Introduction to Solid Modelling using Solidworks

Learning Objectives

- *Understand the need of the sketching environment.*
- *Open a new part document.*
- *Understand various terms used in the sketching environment.*
- *Use various sketching tools.*
- *Use the drawing display tools.*
- *Delete sketched entities.*

THE SKETCHING ENVIRONMENT

A sketch is defined as the basic contour for the feature

Figure 1 Solid model of a Spanner

Figure 2 Base feature of the Spanner

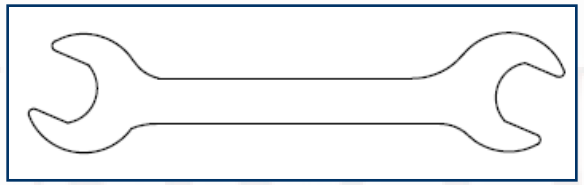

Figure 3 Sketch for the base feature of the Spanner

THE SKETCHING ENVIRONMENT

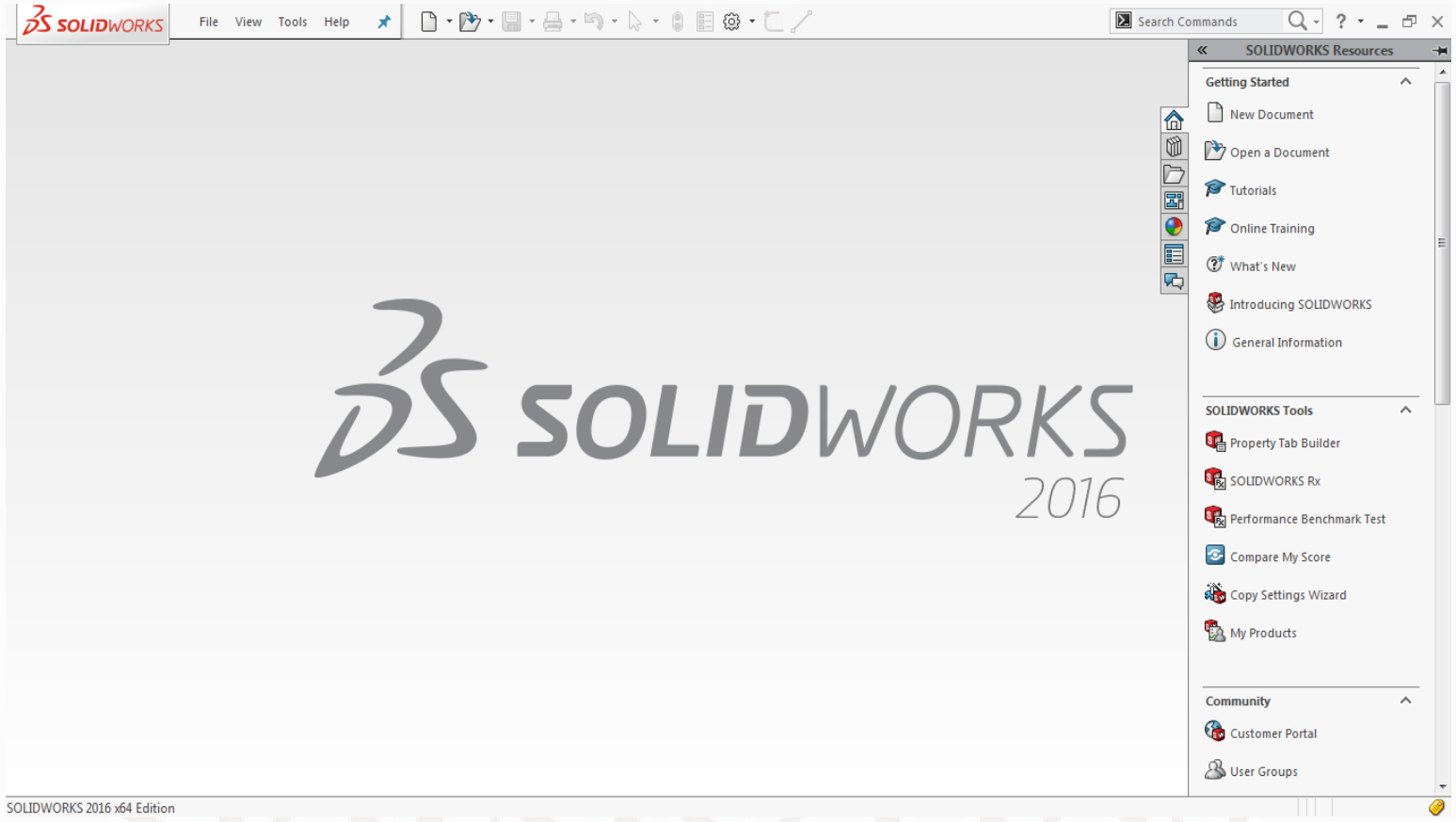

Figure 4 The **SOLIDWORKS** window

THE SKETCHING ENVIRONMENT

➢ TASK PANES

Figure 2-5 Various task panes in SOLIDWORKS

SOLIDWORKS Resources Task Pane

- Getting Started Rollout
- SOLIDWORKS tools Rollout
- Community Rollout
- Online Resources Rollout
- Subscription Services Rollout
- Tip of the Day Message Box

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THE SKETCHING ENVIRONMENT

Design Library Task Pane

File Explorer Task Pane

View Palette Task Pane

Appearance, Scenes, and Decals Task Pane

Custom Properties

THE SKETCHING ENVIRONMENT

➢ STARTING A NEW DOCUMENT IN SOLIDWORKS 2015

- **Part**
- **Assembly**
- **Drawing**

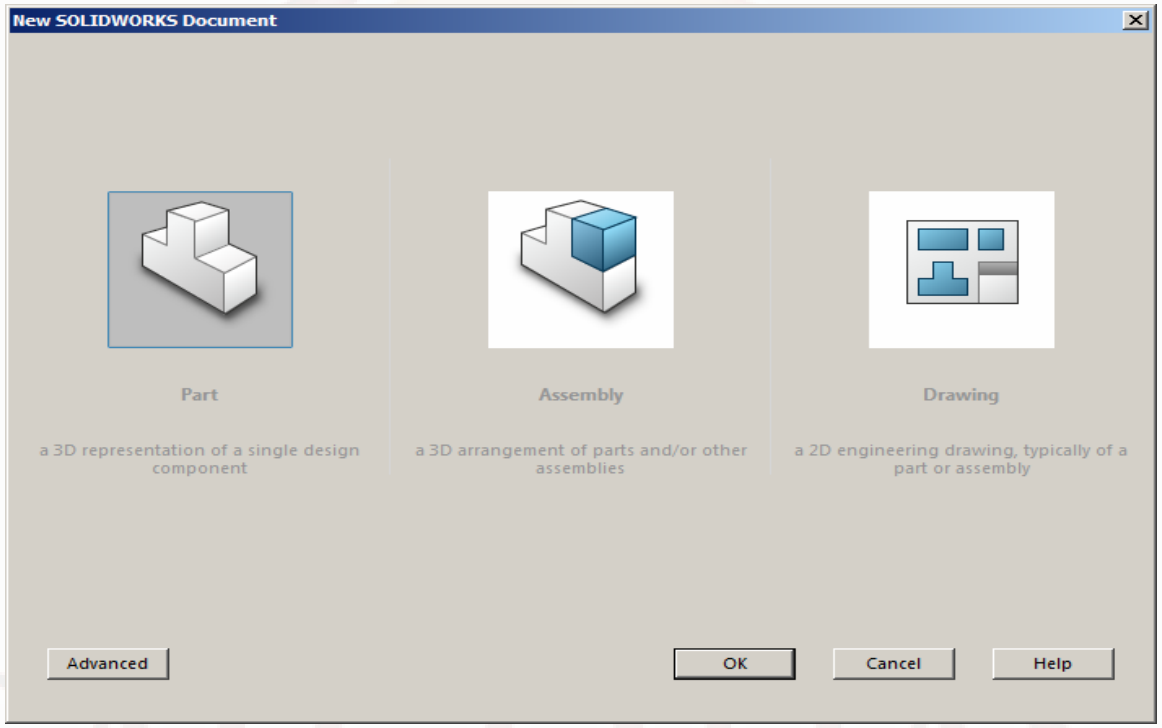

Figure 6 The **New SOLIDWORKS Document** dialog box

THE SKETCHING ENVIRONMENT

➢ UNDERSTANDING THE SKETCHING ENVIRONMENT

Figure 7 shows the Sketch tool in the Menu Bar of SOLIDWORKS 2015

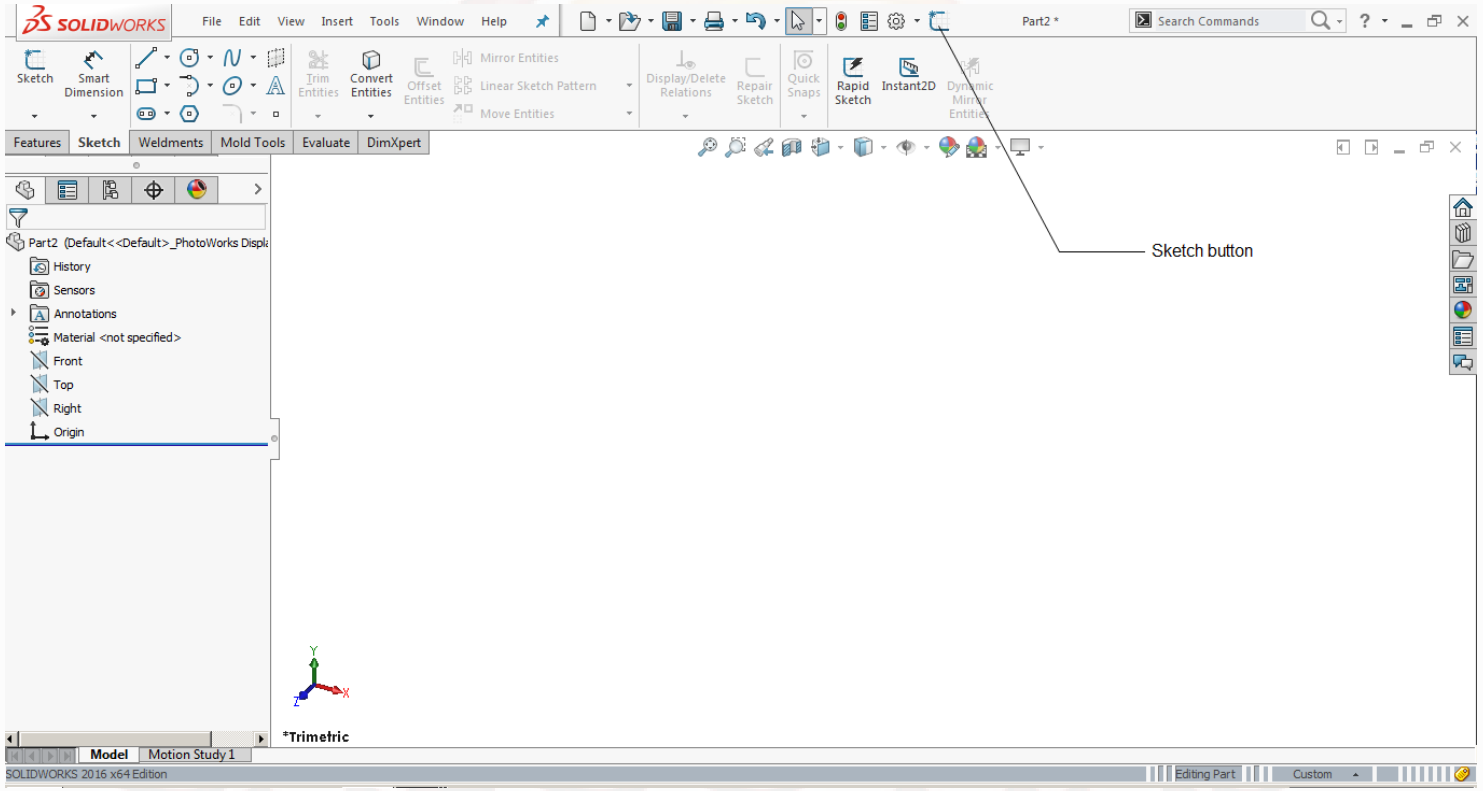

Figure 7 The **Sketch** tool in the Menu Bar of SOLIDWORKS 2016

THE SKETCHING ENVIRONMENT

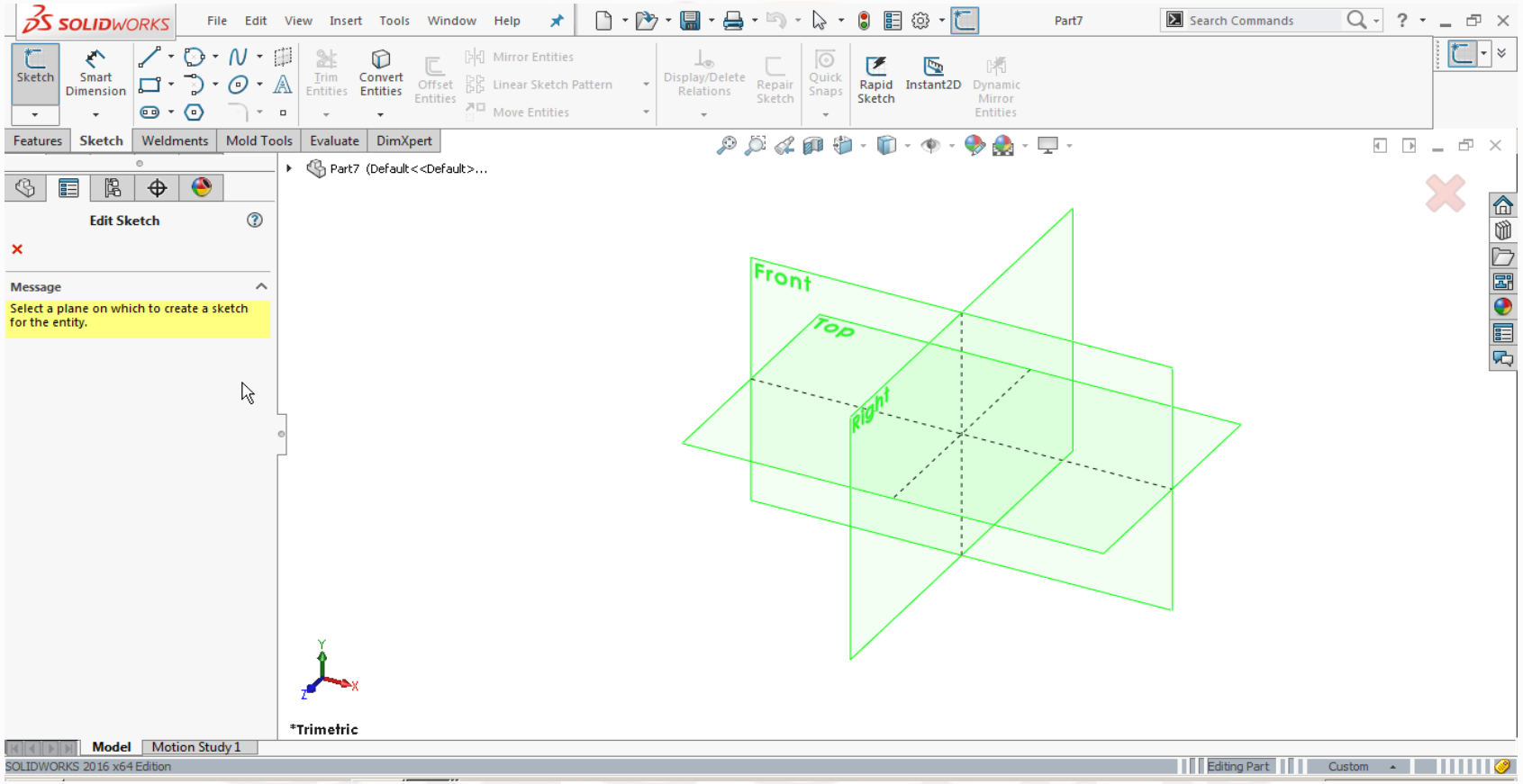

Figure 8 The three default planes displayed on the screen

THE SKETCHING ENVIRONMENT

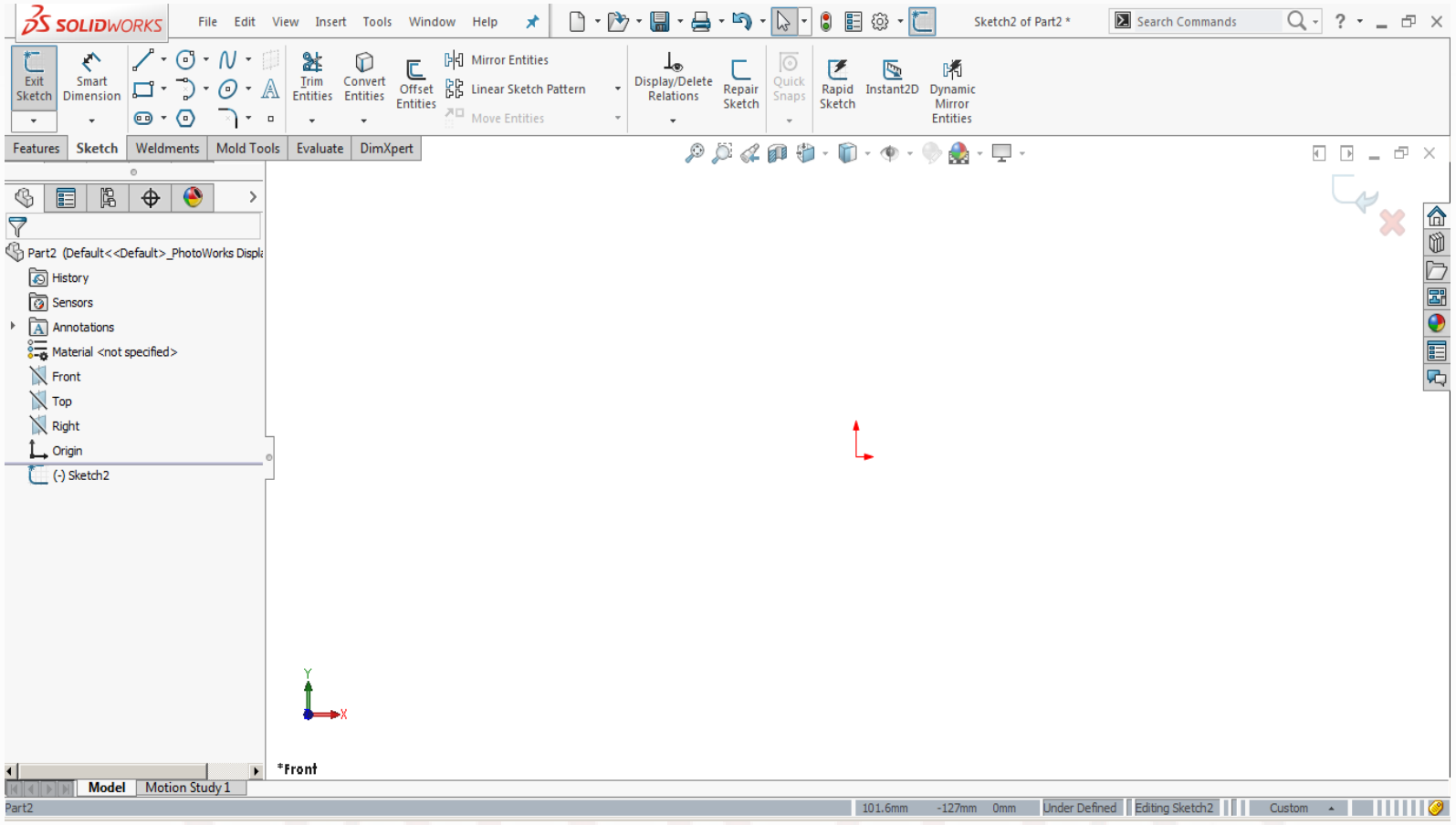

Figure 9 Default screen display of a part document in the sketching environment

THE SKETCHING ENVIRONMENT

➢SETTING THE DOCUMENT OPTIONS

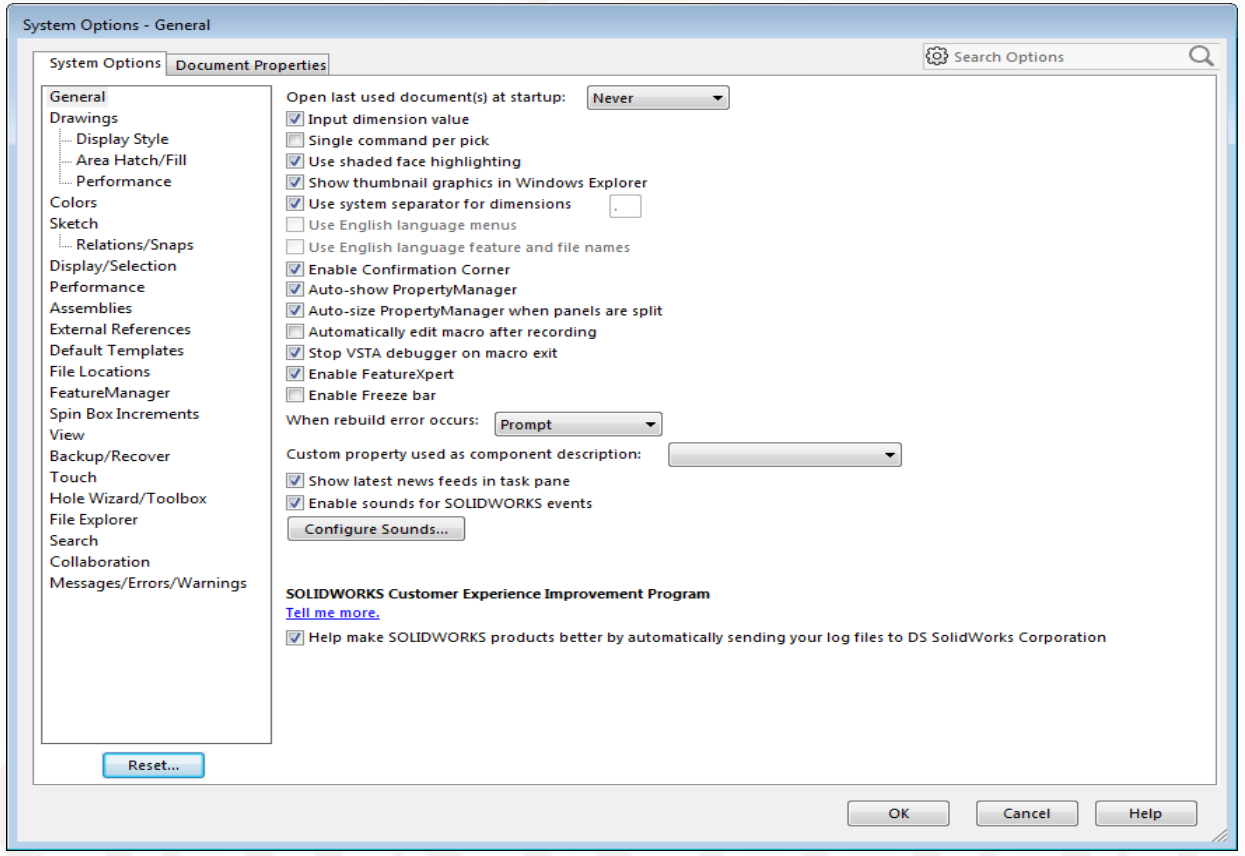

Figure 10 The **System Options - General** dialog box

THE SKETCHING ENVIRONMENT

- **Modifying the Drafting Standards**
- **Modifying the Linear and Angular Units**

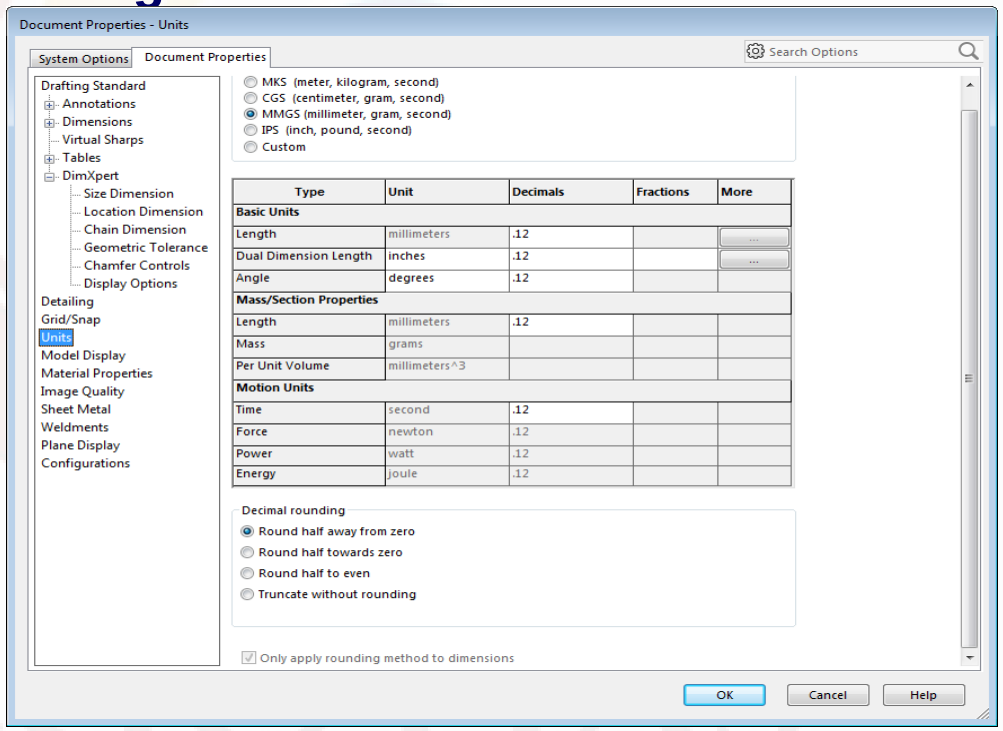

Figure 11 Setting the dimensioning standards

THE SKETCHING ENVIRONMENT

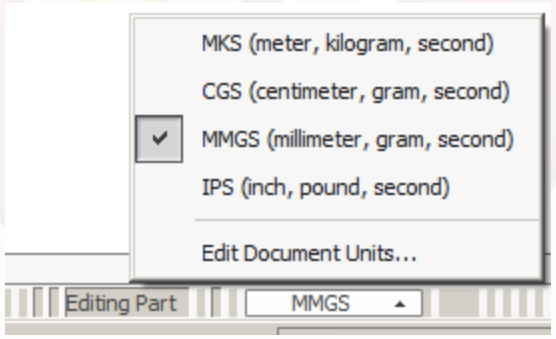

Figure 12 Flyout displayed after choosing the **Unit system** button

THE SKETCHING ENVIRONMENT

- **Modifying the Snap and Grid Settings**

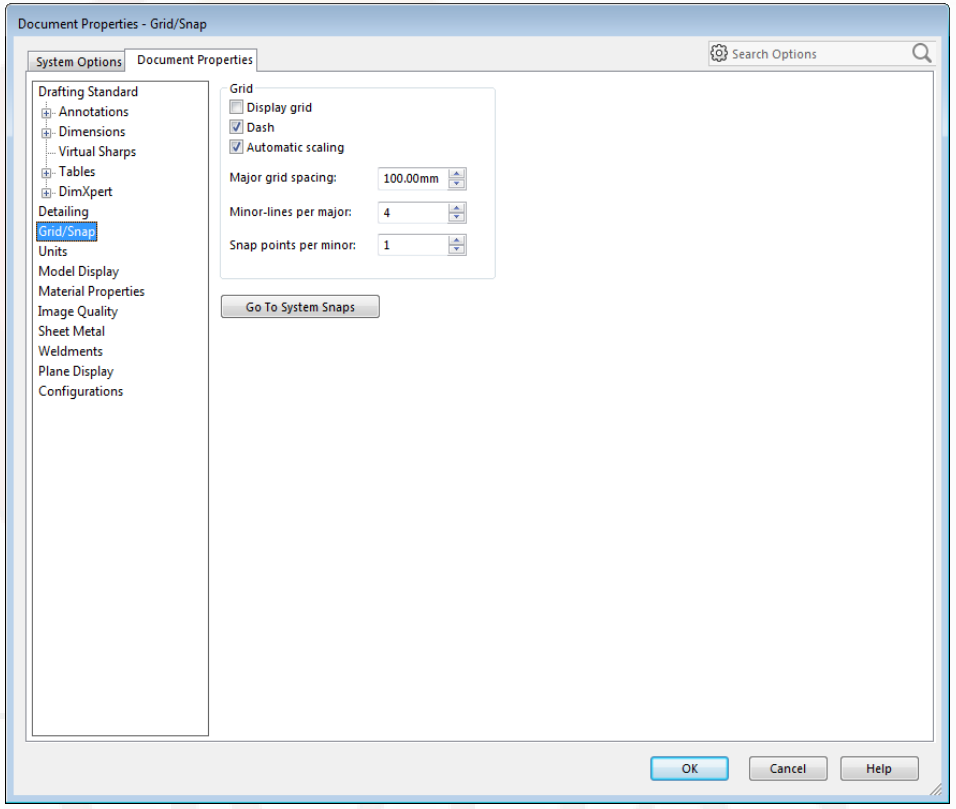

Figure 13 The **Document Properties - Grid/Snap** dialog box

➢ LEARNING SKETCHER TERMS

- **Origin**
- **Inferencing Lines**

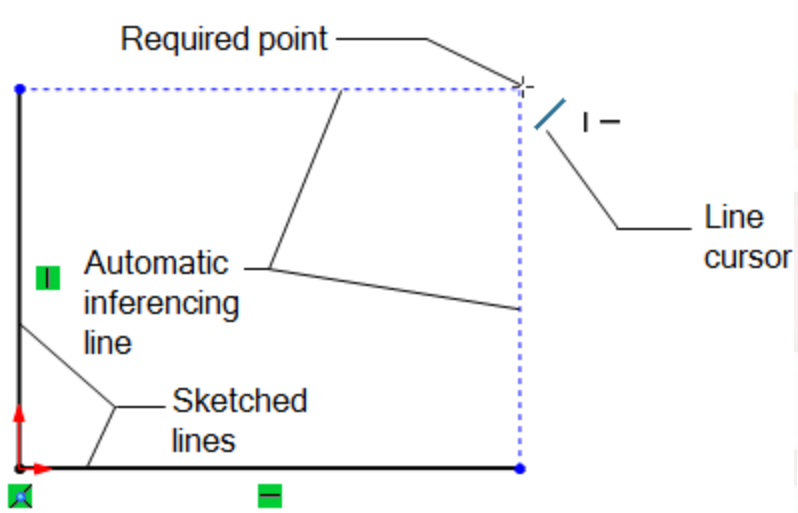

Figure 14 Using inferencing lines to locate a point

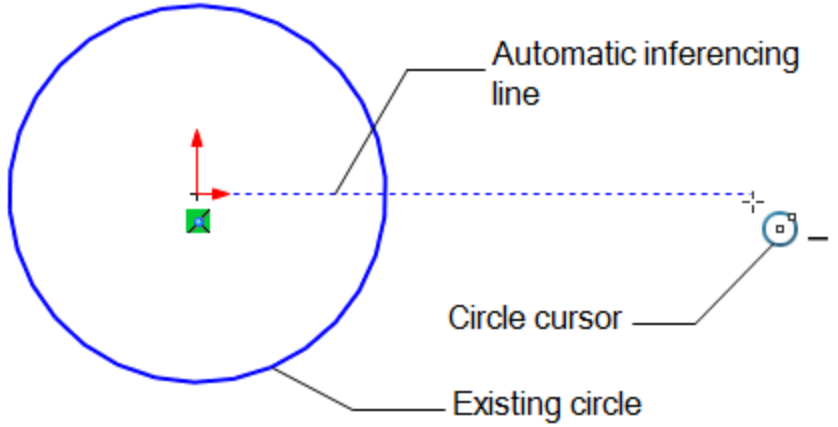

Figure 15 Using inferencing lines to locate the center of a circle

THE SKETCHING ENVIRONMENT

- **Select Tool**

SOLIDWORKS menus: Tools > Select

- **Selecting Entities Using the Box Selection**
- **Selecting Entities Using the Cross Selection**
- **Selecting Entities Using the Lass Selection**
- **Selecting Entities Using the SHIFT and CTRL Keys**
- **Invert Selection Tool**

SOLIDWORKS menus: Tools > Invert Selection
Toolbar: Selection Filter > Invert Selection

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➢ DRAWING LINES

Command Manager: Sketch > Line
SOLIDWORKS menus: Tools > Sketch Entities > Line
Toolbar: Sketch > Line

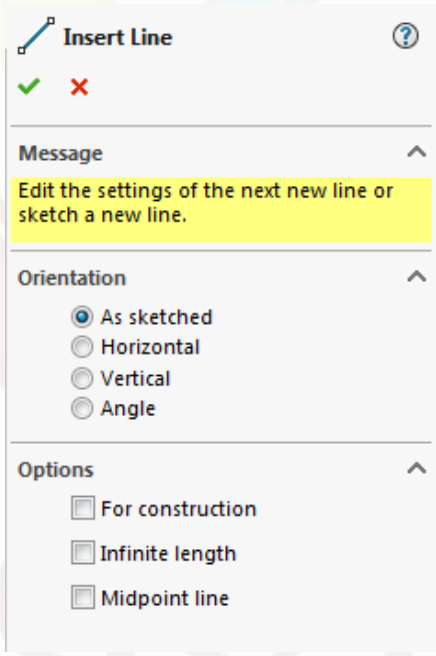

Figure 16 The **Insert Line PropertyManager**

THE SKETCHING ENVIRONMENT

- **Orientation Rollout**
- **Options Rollout**
  - **Drawing a Chain of Continuous Lines**

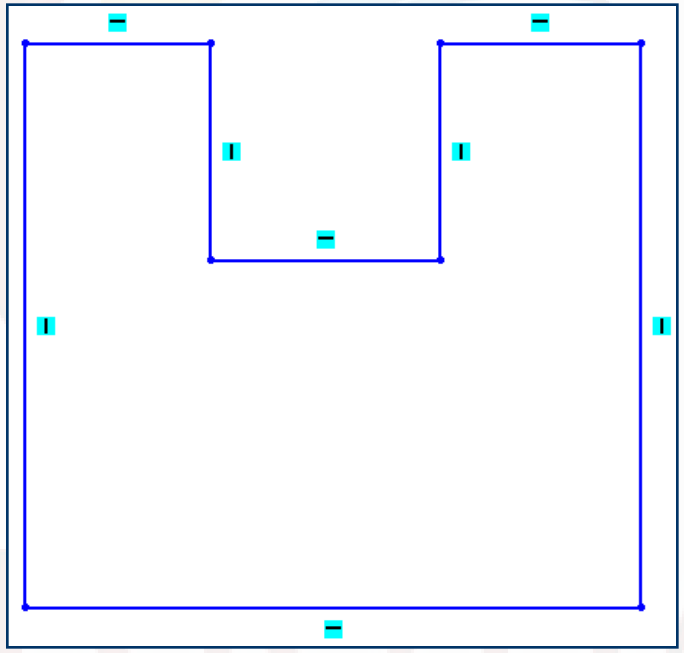

Figure 17 Sketch drawn using continuous lines

THE SKETCHING ENVIRONMENT

- **Drawing Individual Lines**

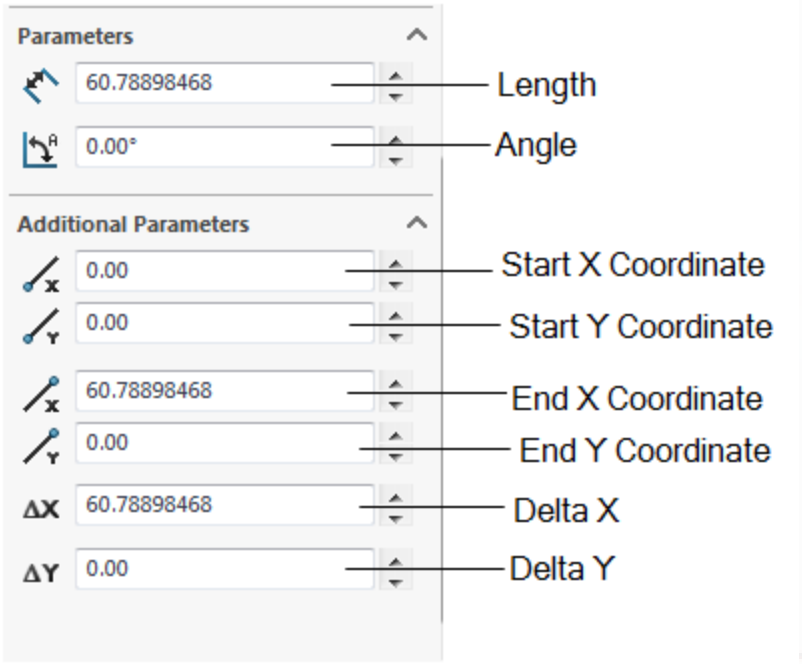

Figure 18 Partial view of the **Line Properties PropertyManager**

- **Line Cursor Parameters**

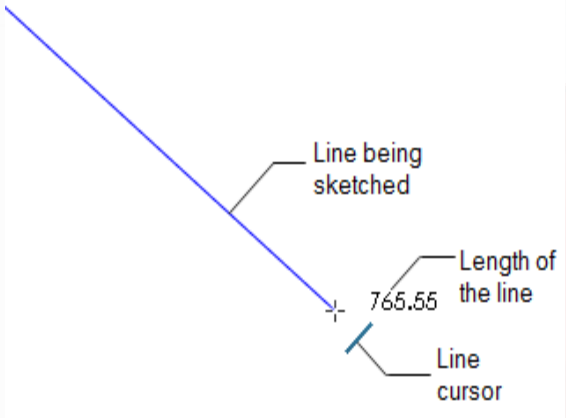

Figure 19 The length of the line displayed on the screen while drawing the line

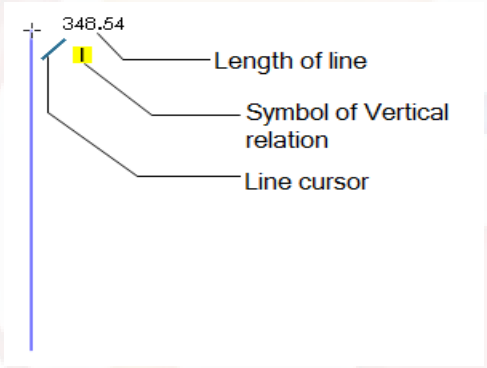

Figure 20 Symbol of the **Vertical** relation

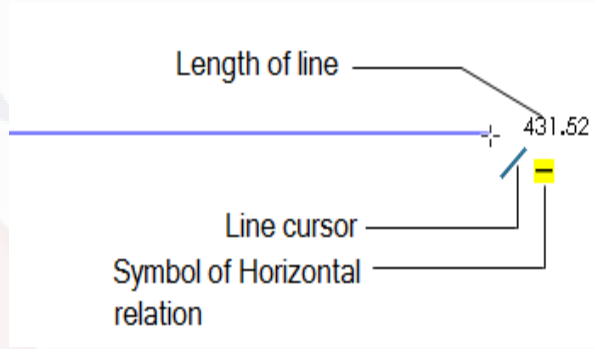

Figure 21 Symbol of the **Horizontal** relation

THE SKETCHING ENVIRONMENT

- **Drawing Tangent or Normal Arcs Using the Line Tool**

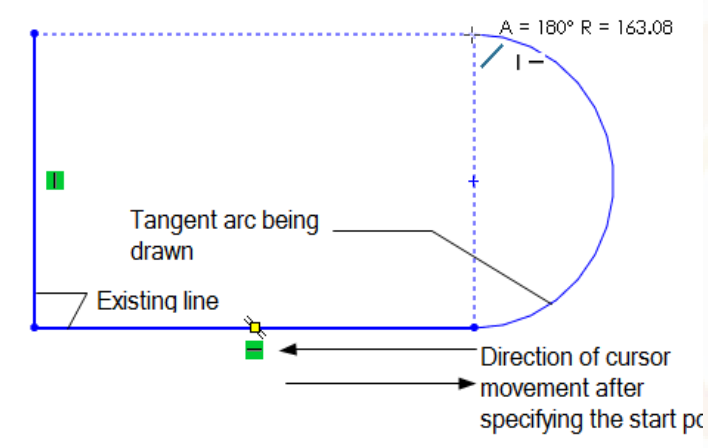

Figure 22 Drawing a tangent arc using the **Line** tool

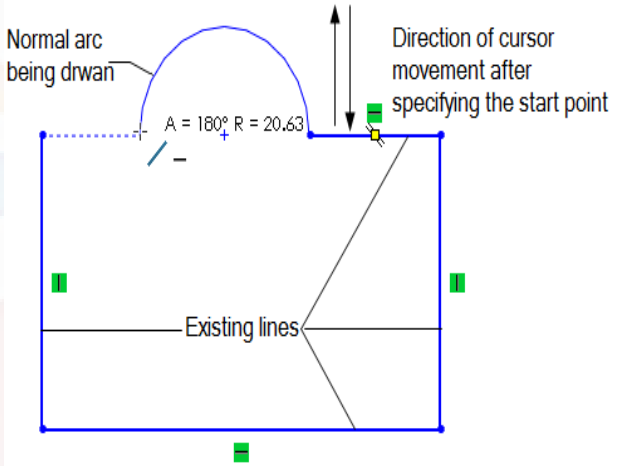

Figure 23 Drawing a normal arc using the **Line** tool

- **Drawing Construction Lines or Centerlines**

Command Manager:	Sketch > Line flyout > Centerline
SOLIDWORKS menus:	Tools > Sketch Entities > Centerline
Toolbar:	Sketch > Line flyout > Centerline

THE SKETCHING ENVIRONMENT

- **Drawing Midpoint Line**

Command Manager:	Sketch > Line flyout > Midpoint Line
SOLIDWORKS menus:	Tools > Sketch Entities > Midpoint Line
Toolbar:	Sketch > Line flyout > Midpoint Line

- **Drawing the Lines of Infinite Length**

➢ DRAWING CIRCLES

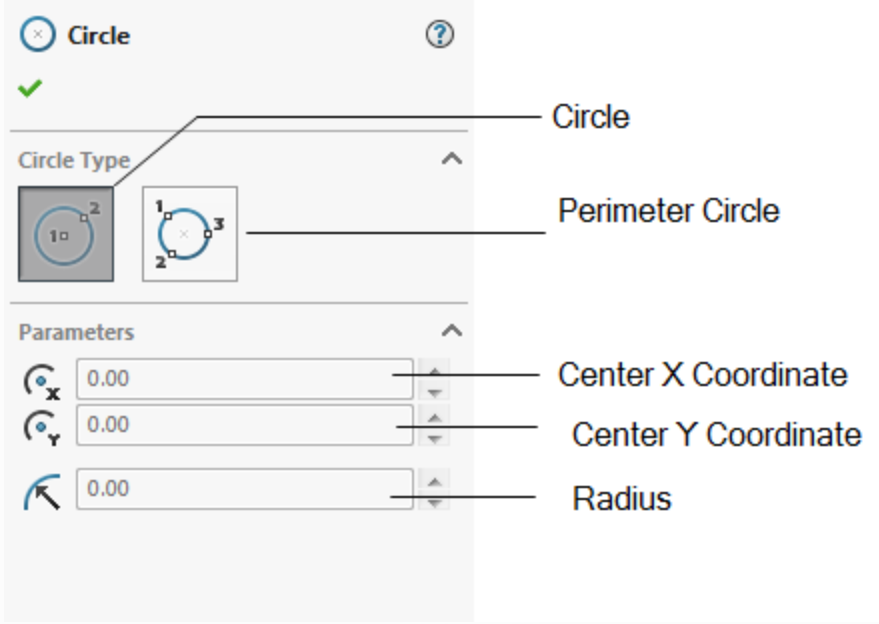

Figure 24 The **Circle PropertyManager**

- **Drawing Circles by Defining Their Center Points**

Command Manager: Sketch > Circle flyout > Circle
SOLIDWORKS menus : Tools > Sketch Entities > Circle
Toolbar: Sketch > Circle

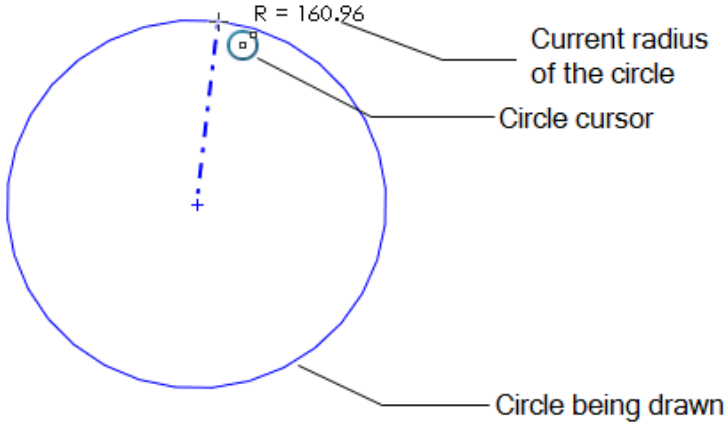

Figure 25 Drawing a circle by specifying the centerpoint

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- **Drawing Circles by Defining Three Points**

Command Manager: Sketch > Circle flyout > Perimeter Circle
SOLIDWORKS menus: Tools > Sketch Entities > Perimeter Circle
Toolbar: Sketch > Circle flyout > Perimeter Circle

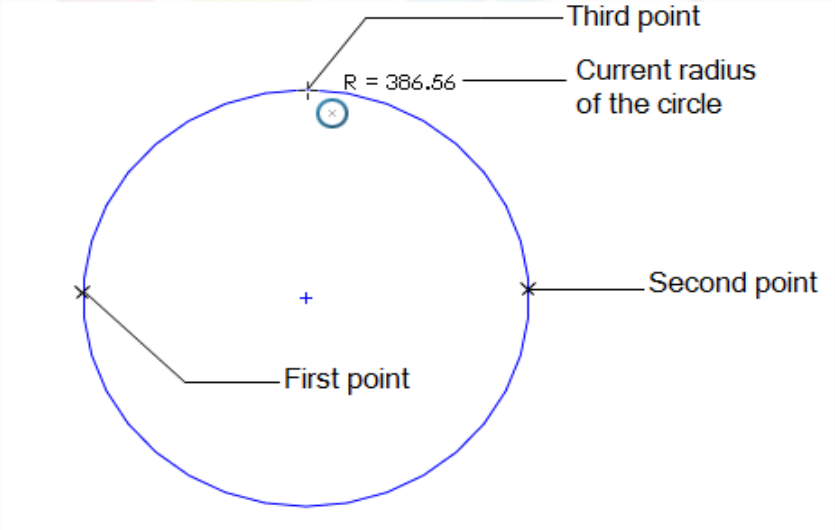

Figure 26 Drawing a circle by specifying three points

- **Drawing Construction Circles**

➢ DRAWING ARCS

- **Drawing Tangent/Normal Arcs**

Command Manager: Sketch > Arc flyout > Tangent Arc
SOLIDWORKS menus: Tools > Sketch Entities > Tangent Arc
Toolbar: Sketch > Arc flyout > Tangent Arc

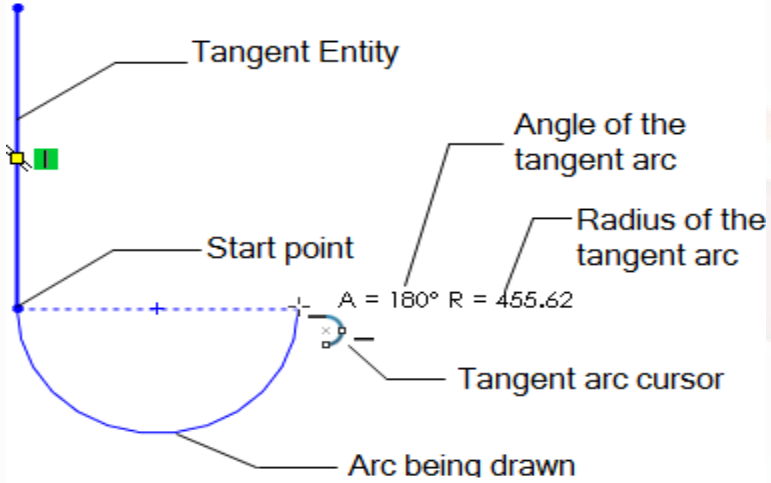

Figure 27 Drawing a tangent arc

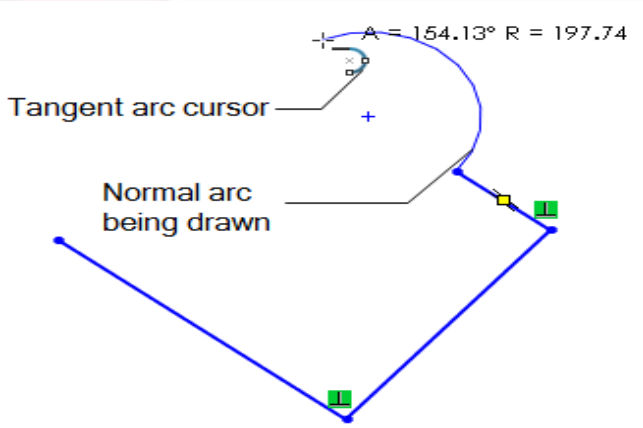

Figure 28 Drawing a normal arc

THE SKETCHING ENVIRONMENT

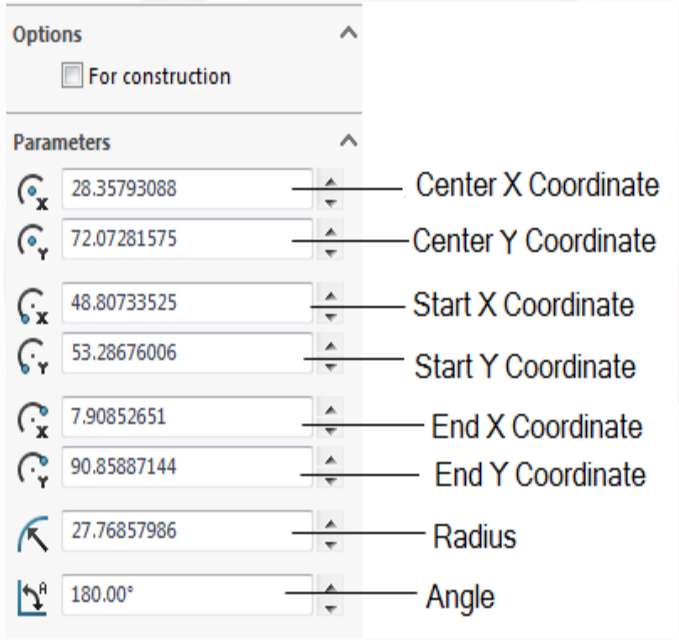

Figure 29 Partial view of the **Arc PropertyManager**

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THE SKETCHING ENVIRONMENT

- **Drawing Centerpoint Arcs**

Command Manager: Sketch > Arc flyout > Centerpoint Arc
SOLIDWORKS menus: Tools > Sketch Entity > Centerpoint Arc
Toolbar: Sketch > Arc flyout > Centerpoint Arc

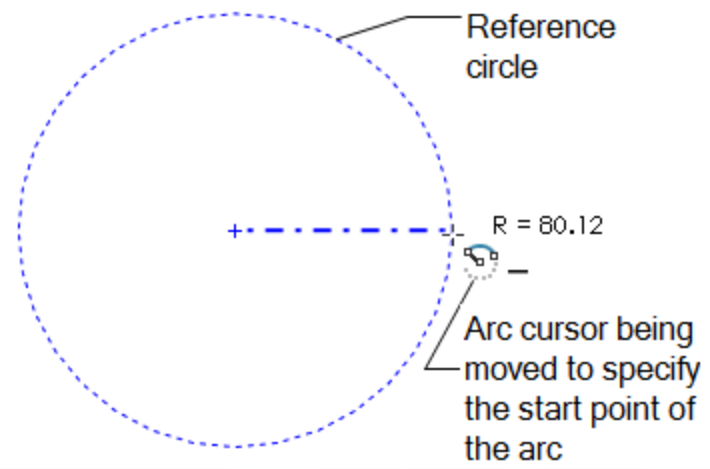

Figure 30 Specifying the Centerpoint and the start point of the Centerpoint arc

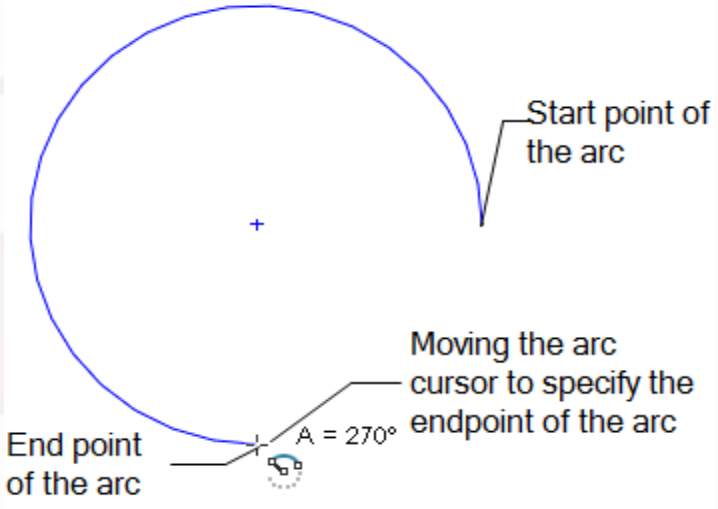

Figure 31 Moving the cursor to specify the start point and the endpoint of the arc

- **Drawing 3 Point Arcs**

Command Manager: Sketch > Arc flyout > 3 Point Arc
SOLIDWORKS menus: Tools > Sketch Entities > 3 Point Arc
Toolbar: Sketch > Arc flyout > 3 Point Arc

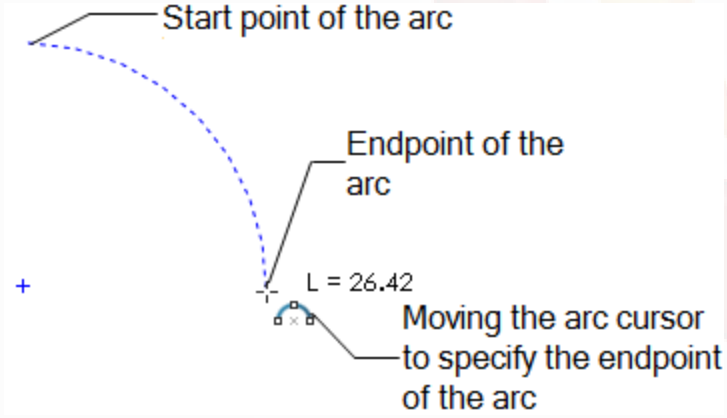

Figure 32 Specifying the start point and the endpoint of the arc

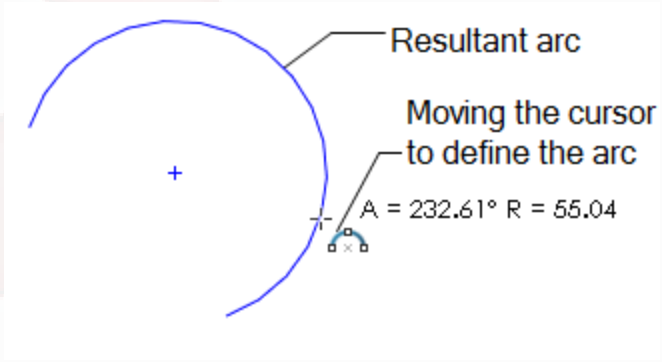

Figure 33 Specifying the third point for drawing the arc

➢ DRAWING RECTANGLES

Tools that are used to draw rectangles are grouped together

- **Drawing Rectangles by Specifying Their Corners**

Command Manager: Sketch > Rectangle flyout > Corner Rectangle
SOLIDWORKS menus: Tools > Sketch Entities > Corner Rectangle
Toolbar: Sketch > Rectangle flyout > Corner Rectangle

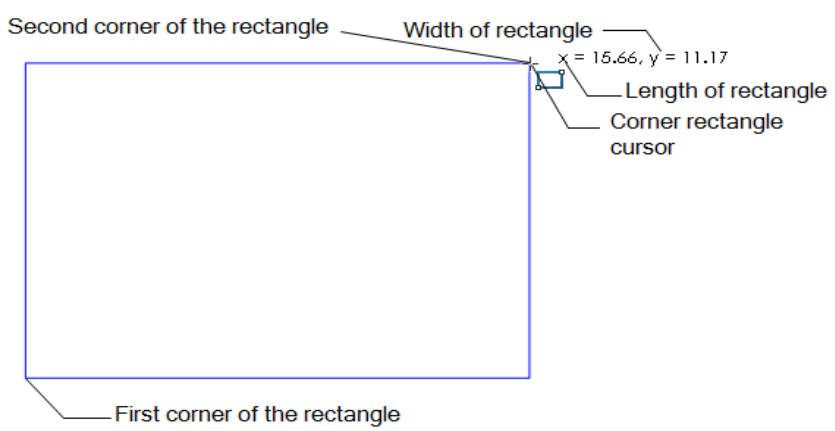

Figure 34 Drawing a rectangle by specifying two diagonally opposite corners

- **Drawing Rectangles by Specifying the Center and a Corner**

Command Manager: Sketch > Rectangle flyout > Center Rectangle
SOLIDWORKS menus: Tools > Sketch Entities > Center Rectangle
Toolbar: Sketch > Rectangle flyout > Center Rectangle

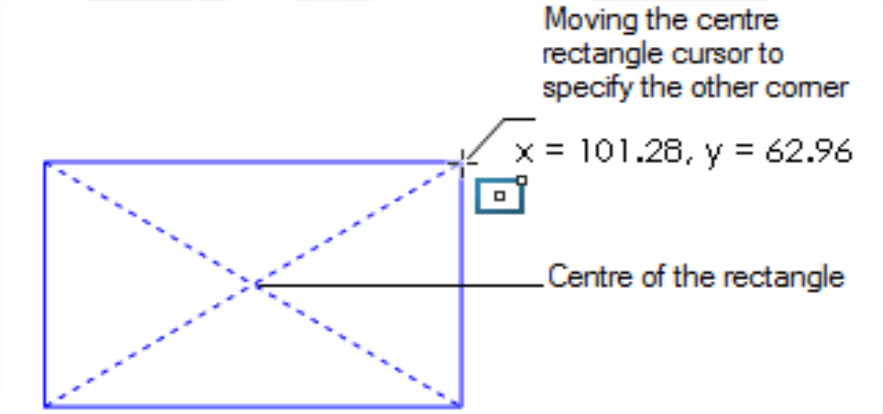

Figure 35 Drawing a rectangle by specifying the center and one of the corners

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- **Drawing Rectangles at an Angle**

Command Manager: Sketch > Rectangle flyout > 3 Point Corner Rectangle

SOLIDWORKS menus: Tools > Sketch Entities > 3 Point Corner Rectangle

Toolbar: Sketch > Rectangle flyout > 3 Point Corner Rectangle

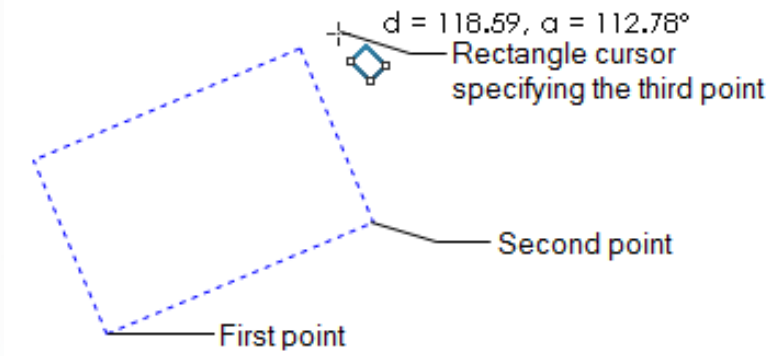

Figure 36 Drawing a rectangle at an angle

THE SKETCHING ENVIRONMENT

• Drawing Centerpoint Rectangles at an Angle

Command Manager: Sketch > Rectangle flyout > 3 Point Center Rectangle

SOLIDWORKS menus: Tools > Sketch Entities > 3 Point Center Rectangle

Toolbar: Sketch > Rectangle flyout > 3 Point Center Rectangle

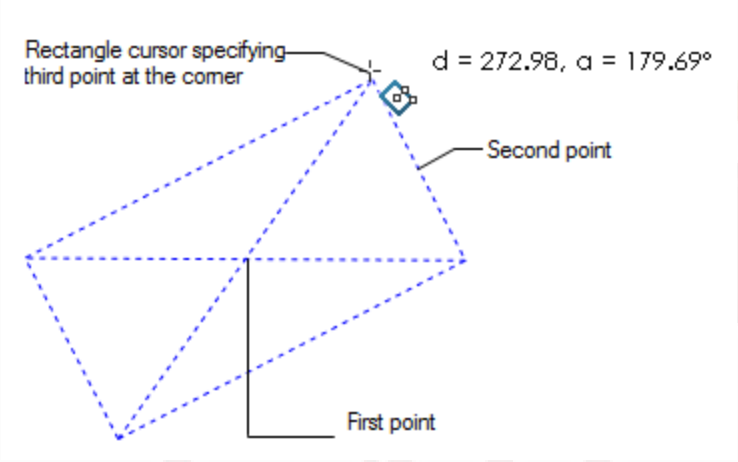

Figure 37 Specifying the third point when the **From corners** radio button is selected

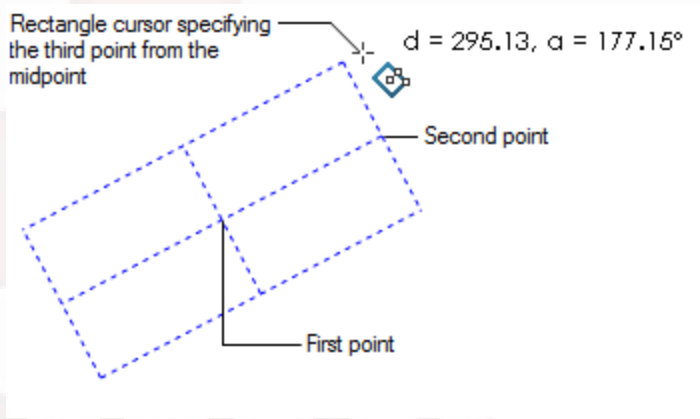

Figure 38 Specifying the third point when the **From Midpoints** radio button is selected

- **Drawing Parallelograms**

Command Manager: Sketch > Rectangle flyout > Parallelogram
SOLIDWORKS menus: Tools > Sketch Entities > Parallelogram
Toolbar: Sketch > Rectangle flyout > Parallelogram

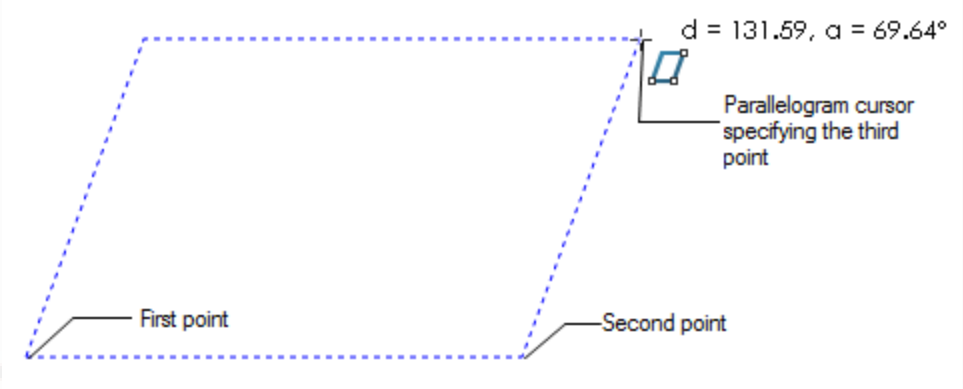

Figure 39 Drawing a parallelogram

➢ DRAWING POLYGONS

Command Manager: Sketch > Polygon
SOLIDWORKS menus: Tools > Sketch Entities > Polygon
Toolbar: Sketch > Polygon

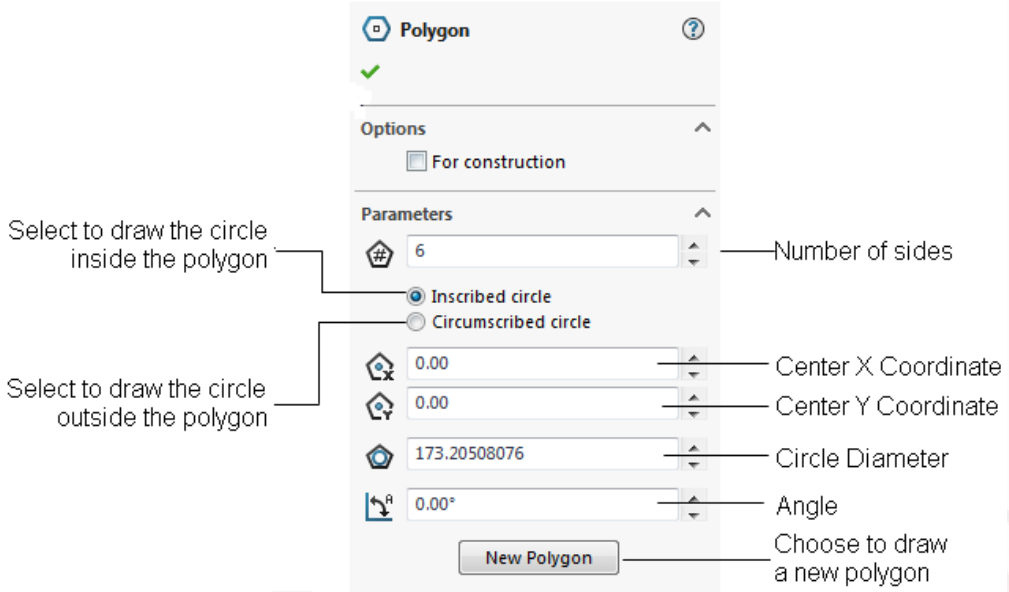

Figure 40 The **Polygon PropertyManager**

THE SKETCHING ENVIRONMENT

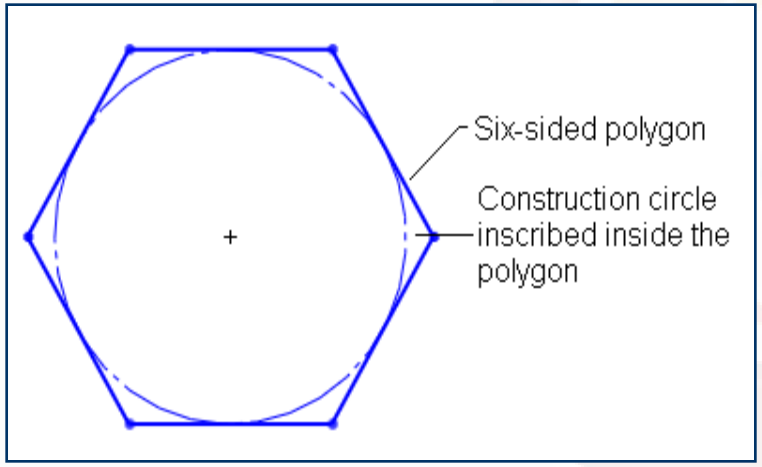

Figure 41 Six-sided polygon with the construction circle inscribed inside it

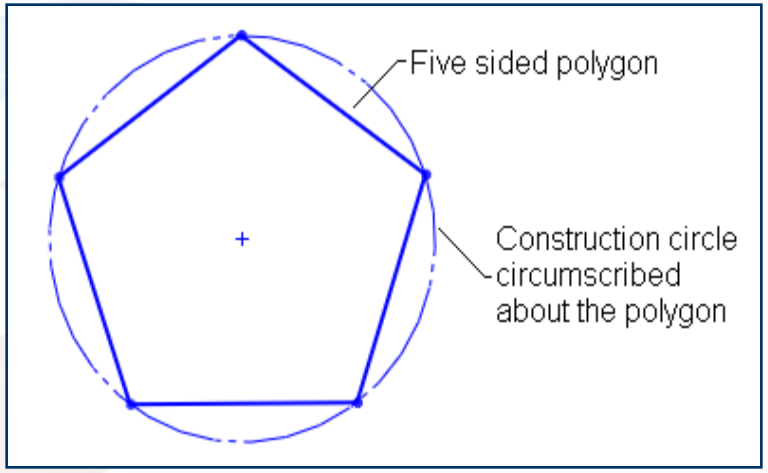

Figure 42 Five-sided polygon with the construction circle circumscribed about it

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➢ DRAWING SPLINES

Command Manager: Sketch > Spline flyout > Spline
SOLIDWORKS menus: Tools > Sketch Entities > Spline
Toolbar: Sketch > Spline flyout > Spline

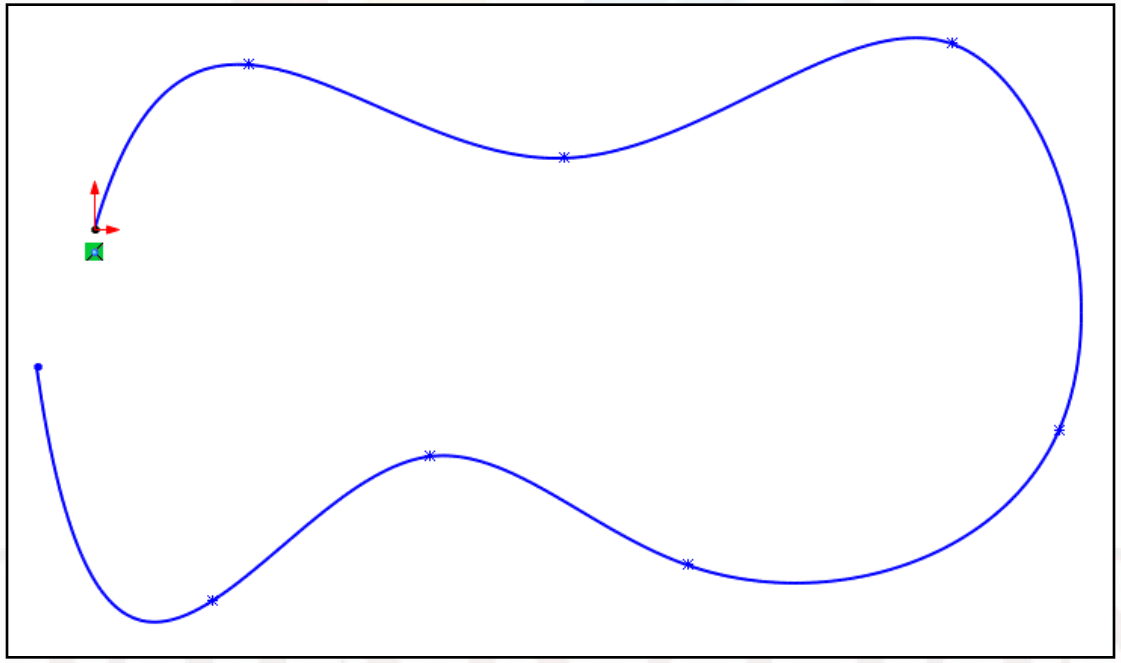

Figure 43 Spline with its start point at the origin

➢ DRAWING SLOTS

Various methods to create draw slots are discussed next

- **Creating a Straight Slot**

Command Manager: Sketch > Slot flyout > Straight Slot
SOLIDWORKS menus: Tools > Sketch Entities > Straight Slot
Toolbar: Sketch > Slot flyout > Straight Slot

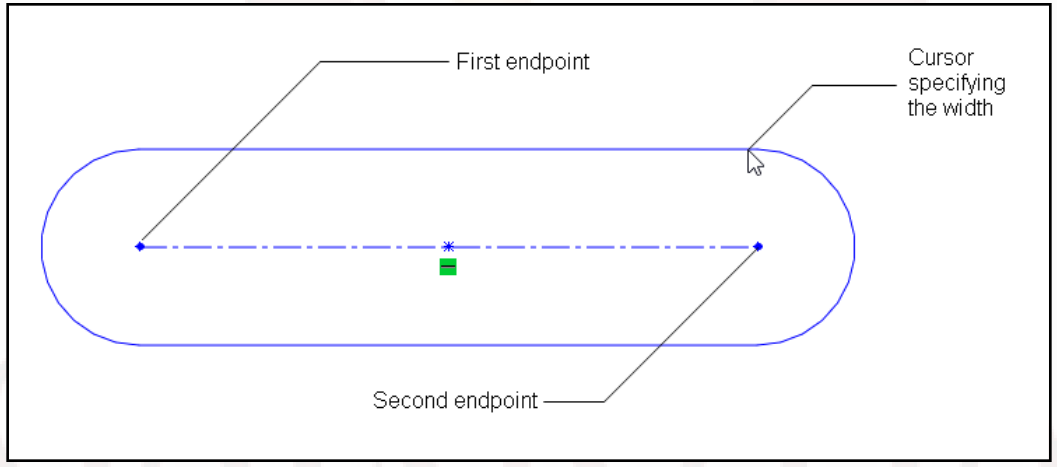

Figure 44 Specifying points to create a straight slot

- **Creating a Centerpoint Straight Slot**

Command Manager: Sketch > Slot flyout > Centerpoint Straight Slot
SOLIDWORKS menus: Tools > Sketch Entities > Centerpoint Straight Slot
Toolbar: Sketch > Slot flyout > Centerpoint Straight Slot

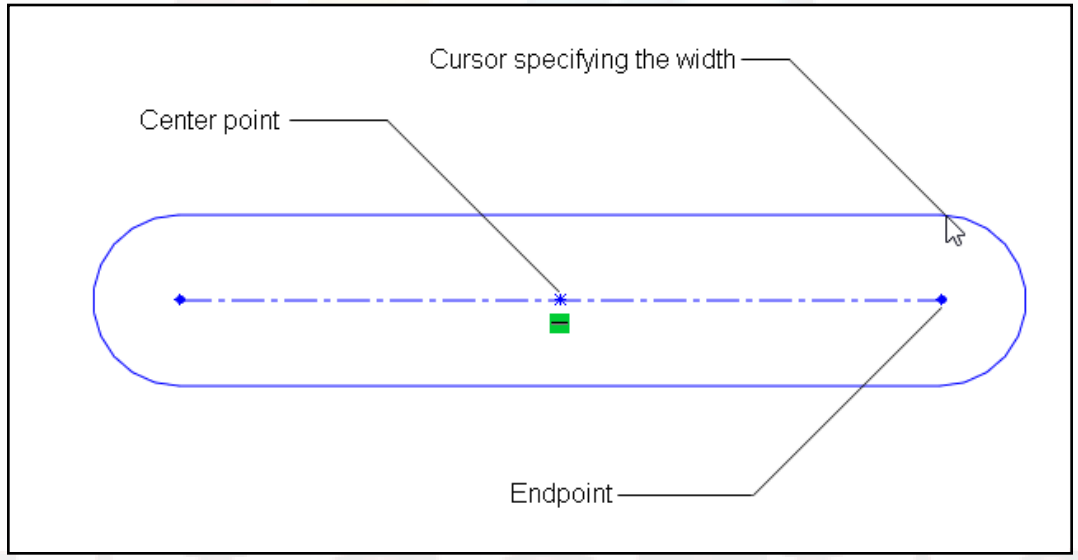

Figure 45 Specifying points to create a centerpoint straight slot

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- **Creating a 3 Point Arc Slot**

Command Manager: Sketch > Slot flyout > 3 Point Arc Slot
SOLIDWORKS menus: Tools > Sketch Entities > 3 Point Arc Slot
Toolbar: Sketch > Slot flyout > 3 Point Arc Slot

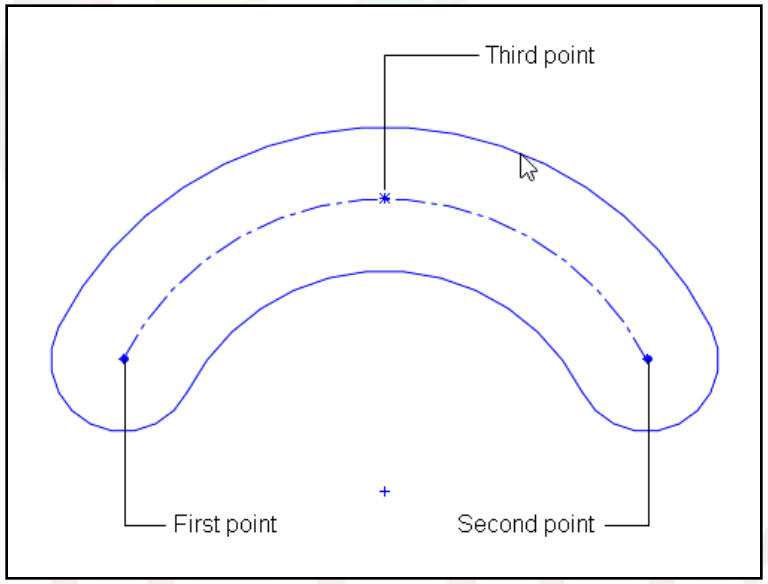

Figure 46 Specifying points to create a 3 point arc slot

- **Creating a Centerpoint Arc Slot**

Command Manager: Sketch > Slot flyout > Centerpoint Arc Slot
SOLIDWORKS menus: Tools > Sketch Entities > Centerpoint Arc Slot
Toolbar: Sketch > Slot Slot > Centerpoint Arc Slot

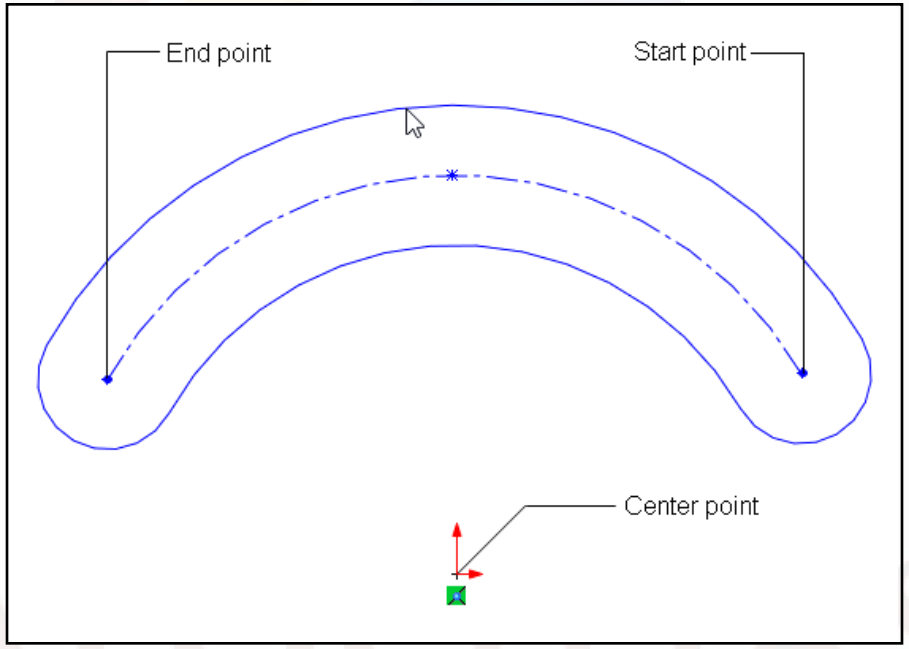

Figure 47 Specifying points to create a centerpoint arc slot

THE SKETCHING ENVIRONMENT

➢ PLACING SKETCHED POINTS

Command Manager: Sketch > Point
SOLIDWORKS menus: Tools > Sketch Entities > Point
Toolbar: Sketch > Point

➢ DRAWING ELLIPSES

Command Manager: Sketch > Ellipse
SOLIDWORKS menus: Tools > Sketch Entities > Ellipse
Toolbar: Sketch > Ellipse

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THE SKETCHING ENVIRONMENT

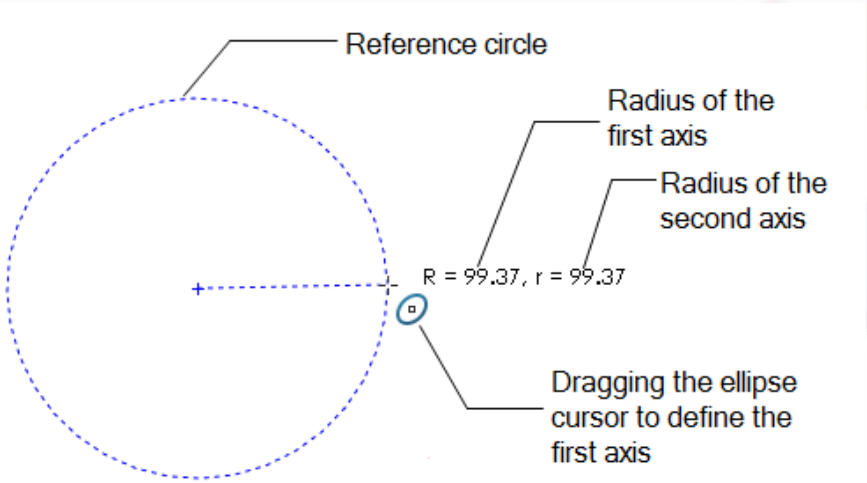

Figure 48 Dragging the cursor to define the ellipse axis

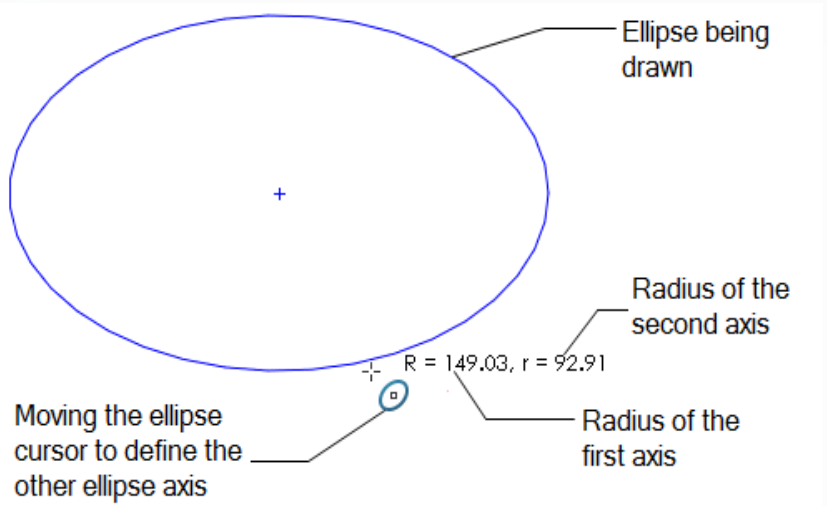

Figure 49 Defining the second axis of the ellipse

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➢ DRAWING ELLIPTICAL ARCS

Command Manager: Sketch > Ellipse flyout > Partial Ellipse
SOLIDWORKS menus: Tools > Sketch Entities > Partial Ellipse
Toolbar: Sketch > Ellipse flyout > Partial Ellipse

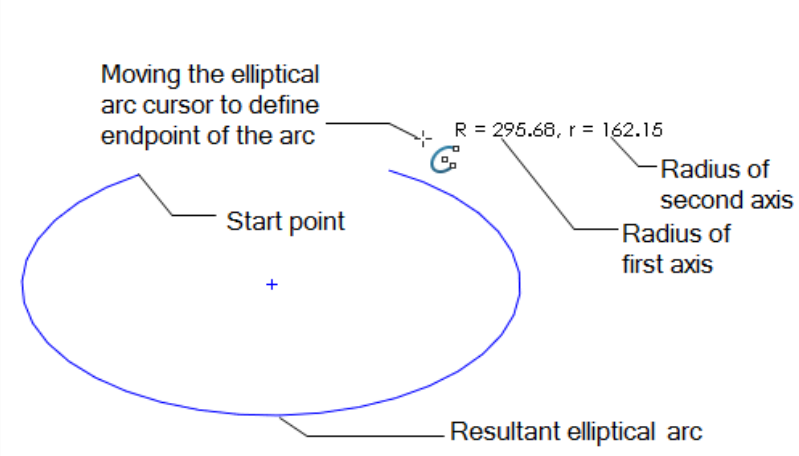

Figure 50 Drawing an elliptical arc

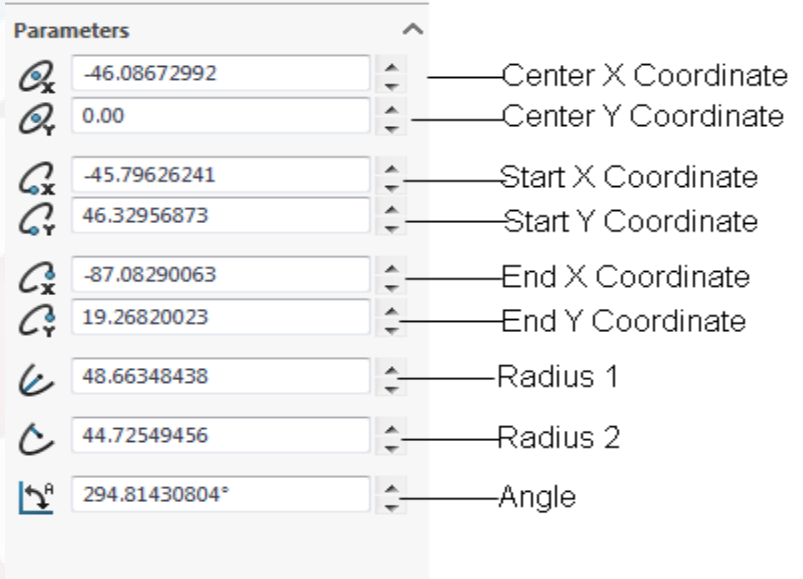

Figure 51 Partial view of the **Ellipse PropertyManager**

➢ DRAWING PARABOLIC CURVES

Command Manager: Sketch > Ellipse flyout > Parabola
SOLIDWORKS menus: Tools > Sketch Entities > Parabola
Toolbar: Sketch > Ellipse flyout > Parabola

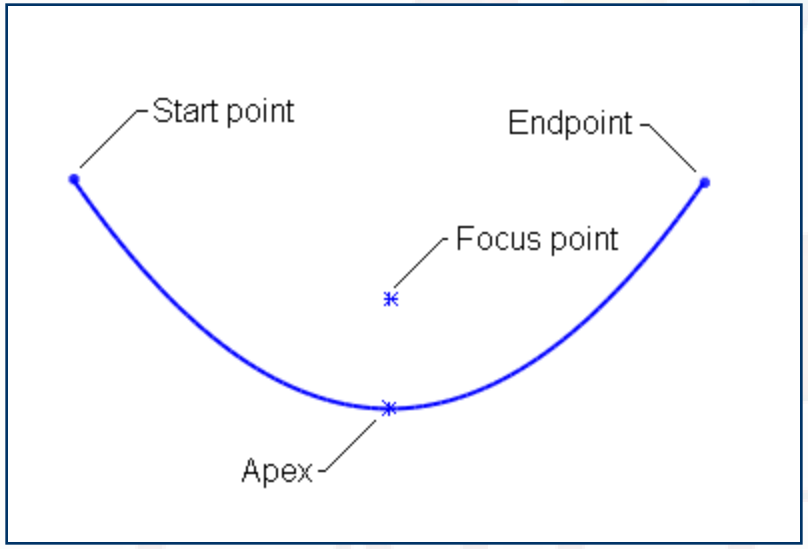

Figure 52 Parabola and its parameters

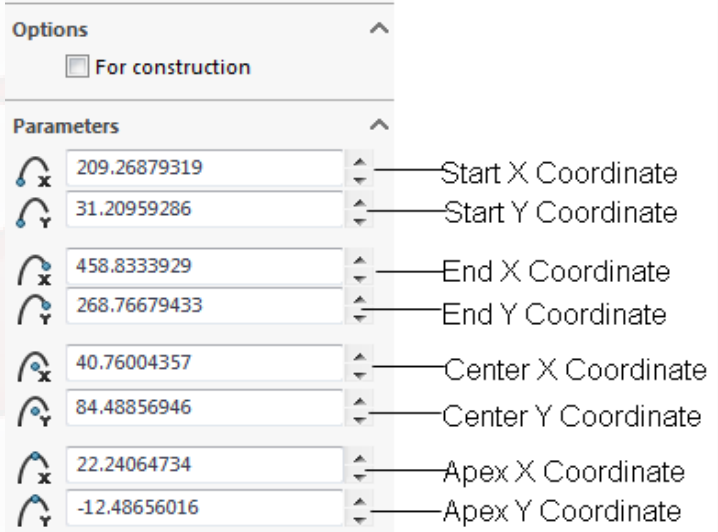

Figure 53 Partial view of the **Parabola PropertyManager**

➢ DRAWING CONIC CURVES

Command Manager: Sketch > Ellipse flyout > Conic
SOLIDWORKS menus: Tools > Sketch Entities >Conic
Toolbar: Sketch > Ellipse flyout > Conic

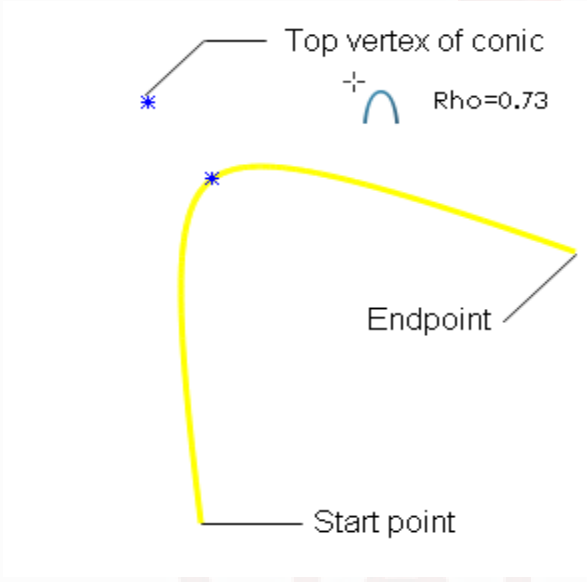

Figure 54 Conic and its parameters

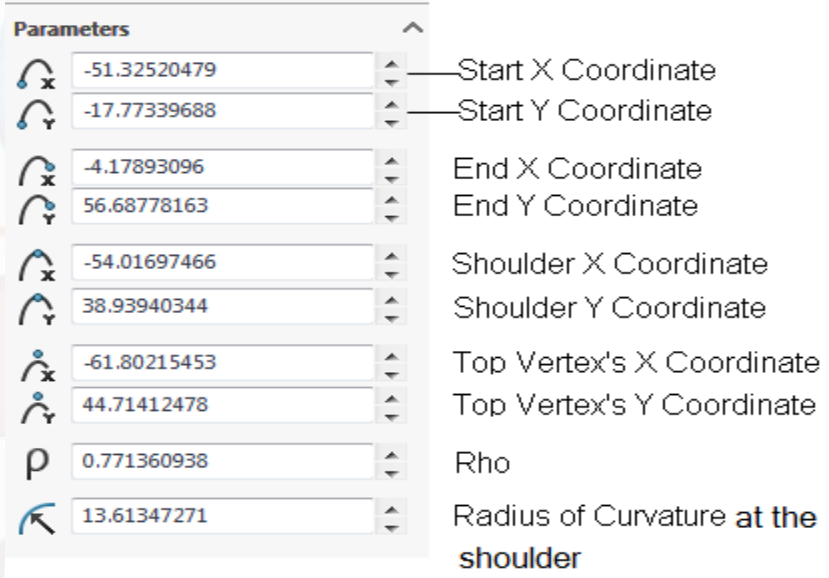

Figure 55 The **Conic PropertyManager**

THE SKETCHING ENVIRONMENT

➢ DRAWING DISPLAY TOOLS

Allows you to modify the display of a drawing by zooming or panning it

- **Zoom to Fit**

View (Heads-Up): Zoom to Fit

- **Zoom to Area**

View (Heads-Up): Zoom to Area

- **Zoom In/Out**

SolidWorks menus: View > Modify > Zoom In/Out

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THE SKETCHING ENVIRONMENT

- **Zoom to Selection**

  SOLIDWORKS menus: View > Modify > Zoom to Selection

- **Pan**

  View (Heads-Up): Pan (Customize to Add)

- **Previous View**

  View (Heads-Up): View > Previous View

- **Redraw**

  SOLIDWORKS menus: View > Redraw

➢ DELETING SKETCHED ENTITIES

Sketched entities can be deleted by selecting them using the Select tool and then pressing the DELETE key on the keyboard

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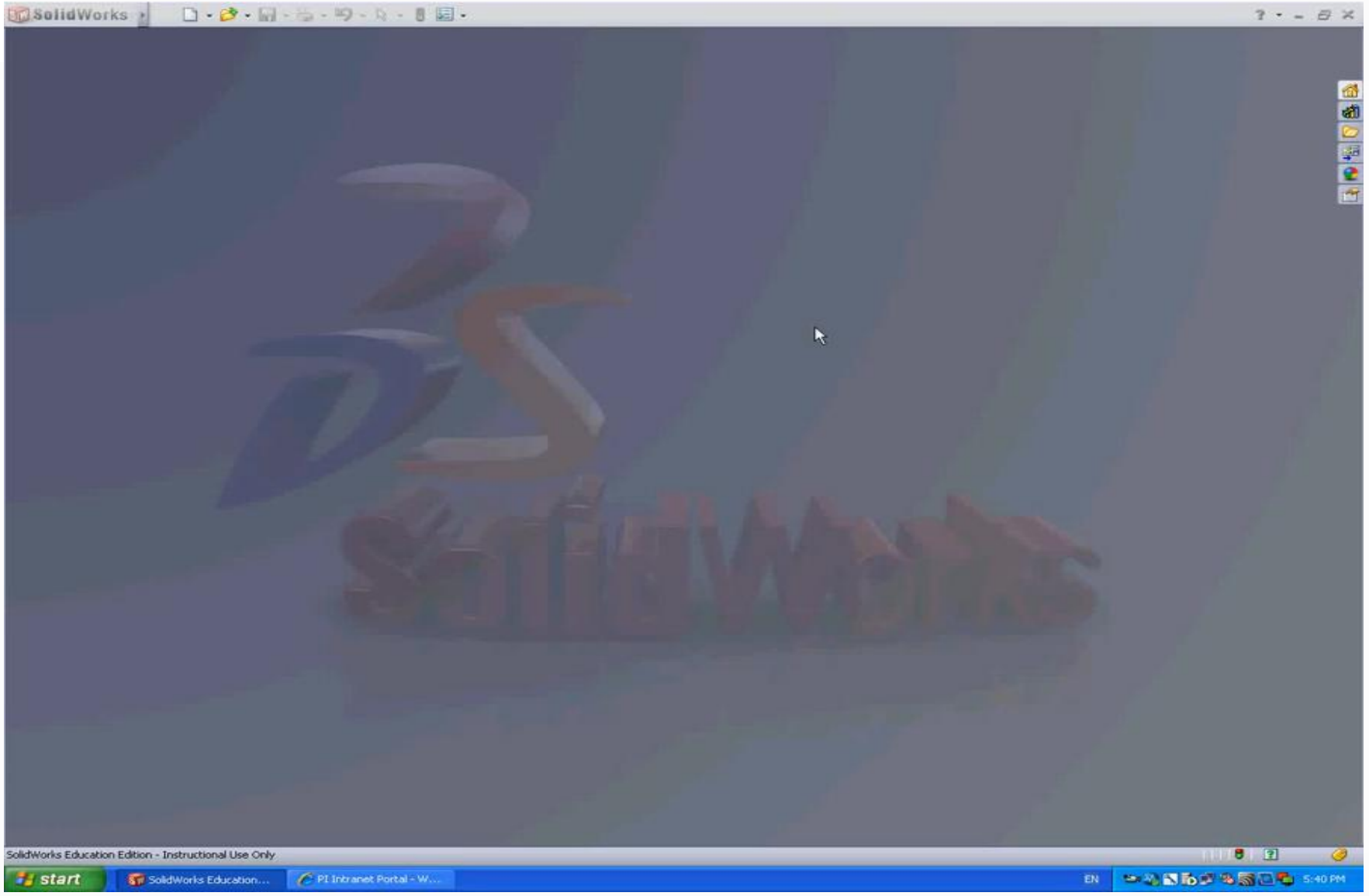
Software Demonstration



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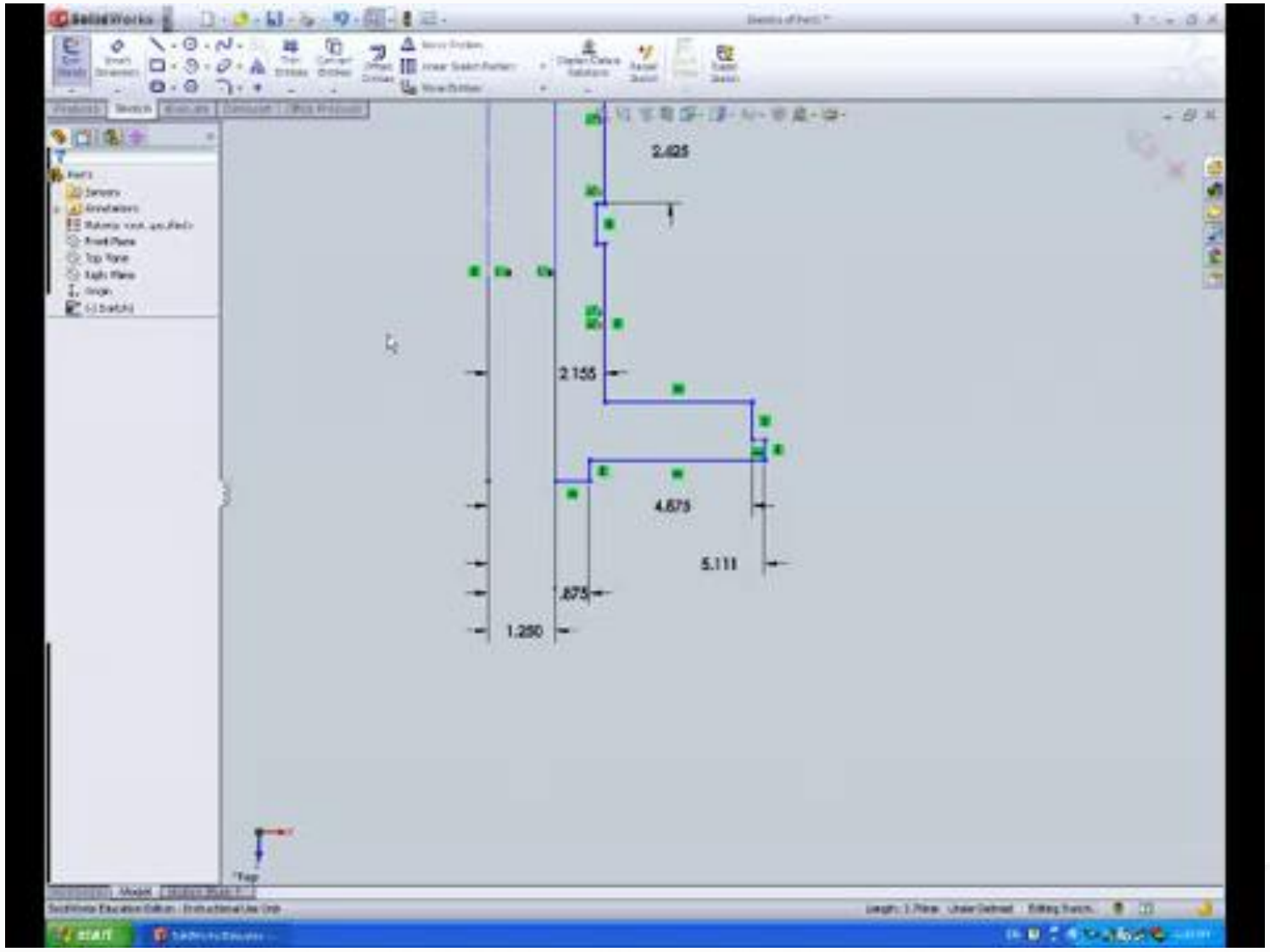
Software Demonstration



Tutorial for practice

➢ Tutorial 1

In this tutorial, you will draw the basic sketch of the revolved solid model shown in **Figure 56**. The sketch of this model is shown in **Figure 57**. Do not dimension the sketch. The solid model and its dimensions are given for your reference only.

(Expected time: 30 min)

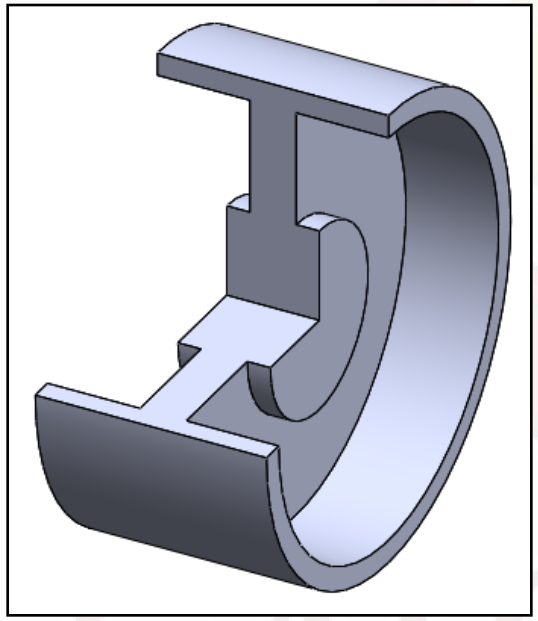


Figure 56 Revolved solid model for Tutorial 1

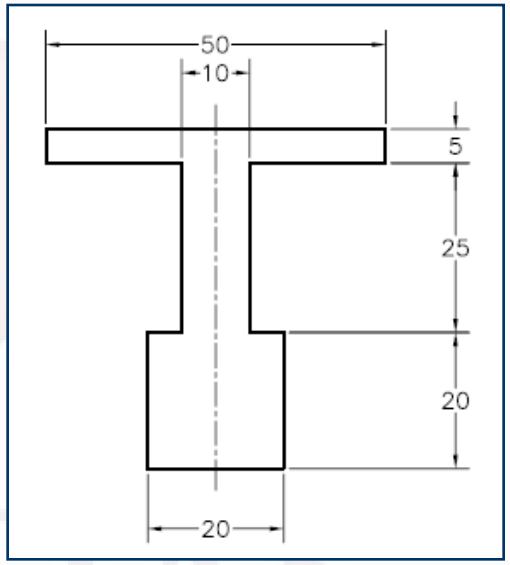


Figure 57 Sketch for the revolved solid model

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Tutorial for practice

The following steps are required to complete this tutorial:

1. Start a new part document.
2. Switch to the sketching environment, refer to **Figures 58** and **59**.

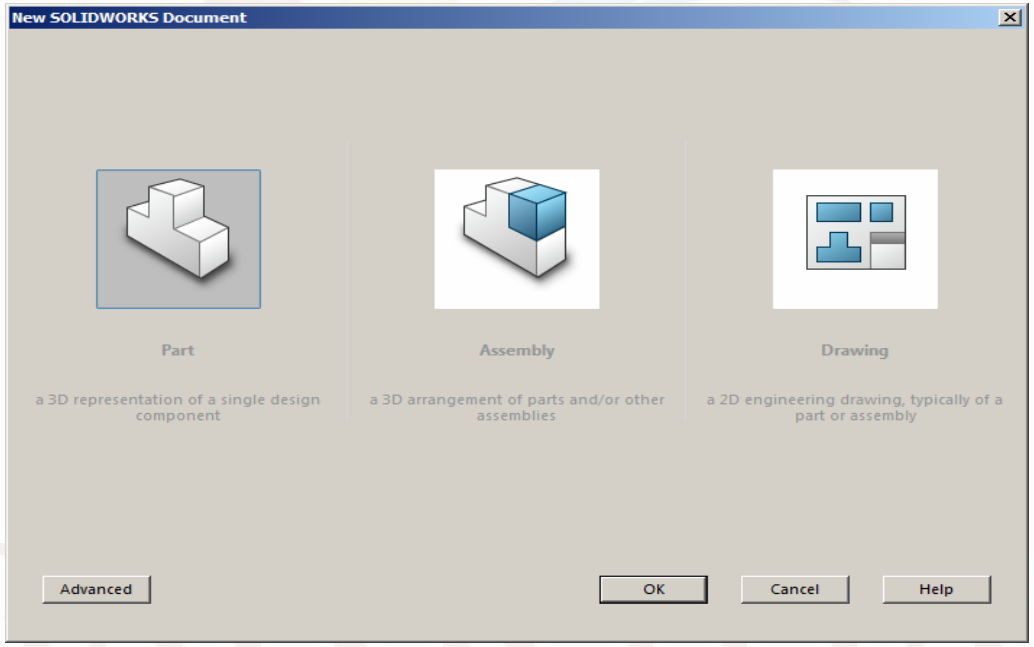

Figure 58 The **New SOLIDWORKS Document** dialog box

Tutorial for practice

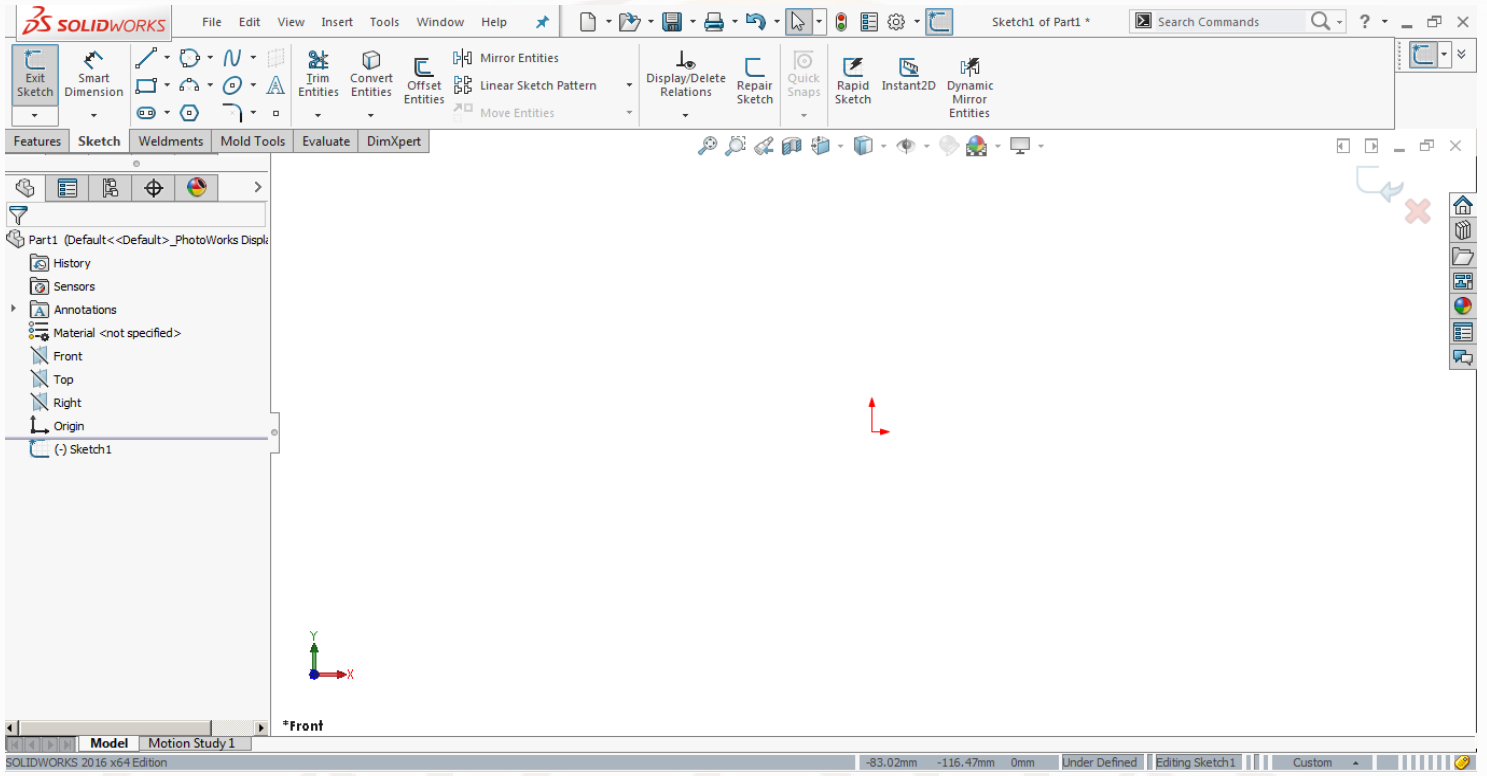

Figure 59 Screen display in the sketching environment

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Tutorial for practice

3. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5 mm instead of 10 mm.

4. Draw the sketch of the model using the **Line** tool, refer to **Figure 60**.

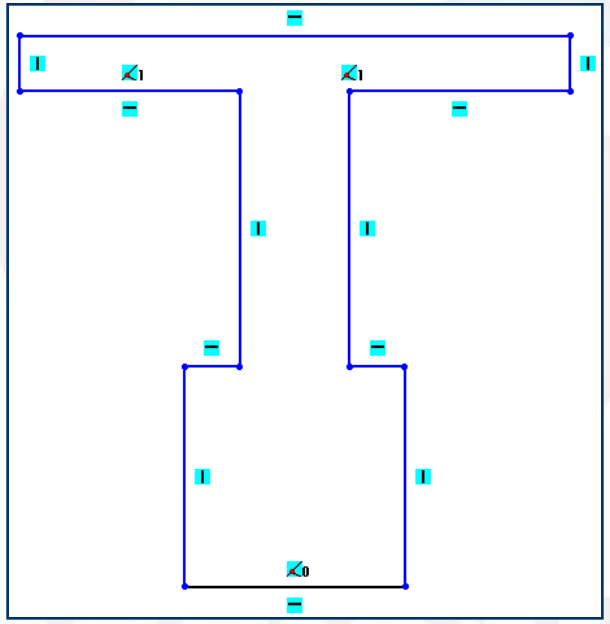

Figure 60 Final sketch for Tutorial 1

5. Save the sketch and then close the document.

➢ Tutorial 2

In this tutorial, you will draw the sketch of the solid model shown in **Figure 61**. The sketch of the model is shown in **Figure 62**. Do not dimension the sketch. The solid model and the dimensions are given for your reference only. **(Expected time: 30 min)**

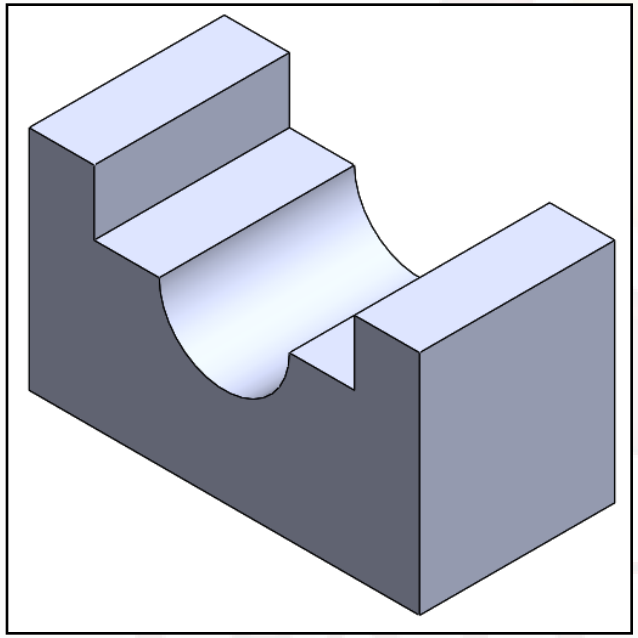

Figure 61 Solid model for Tutorial 2

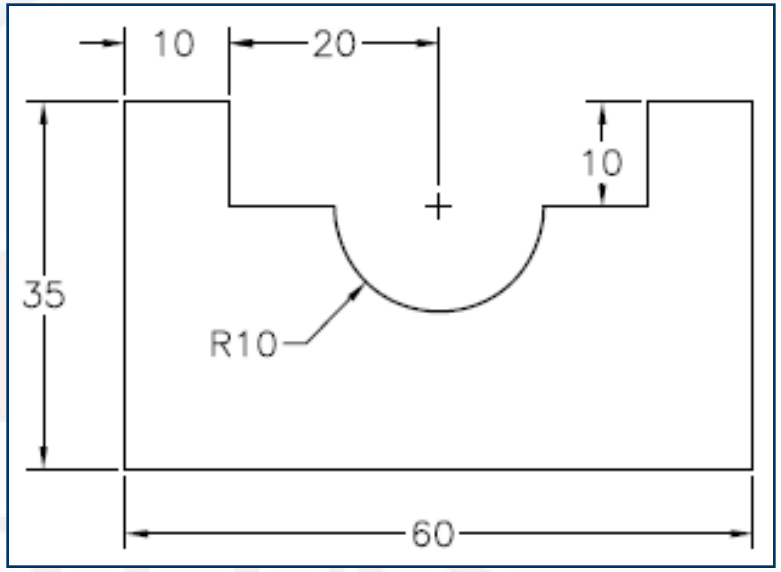

Figure 62 Sketch for Tutorial 2

Tutorial for practice

The following steps are required to complete this tutorial:

1. Start SOLIDWORKS and then start a new part document.
2. Invoke the sketching environment.
3. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5 mm instead of 10 mm.
4. Draw the sketch using the **Line** tool, refer to **Figure 63**.

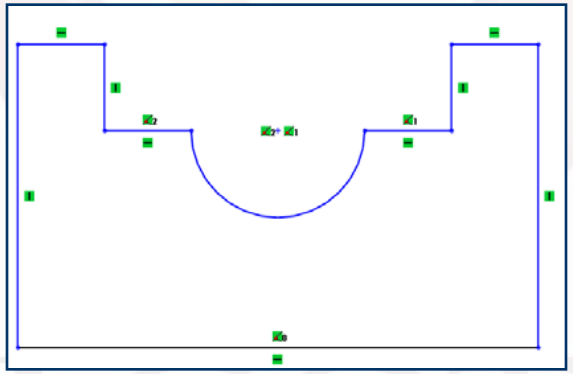

Figure 63 Final sketch for Tutorial 2

5. Save the sketch and then close the file.

➢ Tutorial 3

In this tutorial, you will draw the basic sketch of the model shown in **Figure 64**. The sketch to be drawn is shown in **Figure 65**. Do not dimension the sketch; the solid model and its dimensions are given for your reference only.

(Expected time: 30 min)

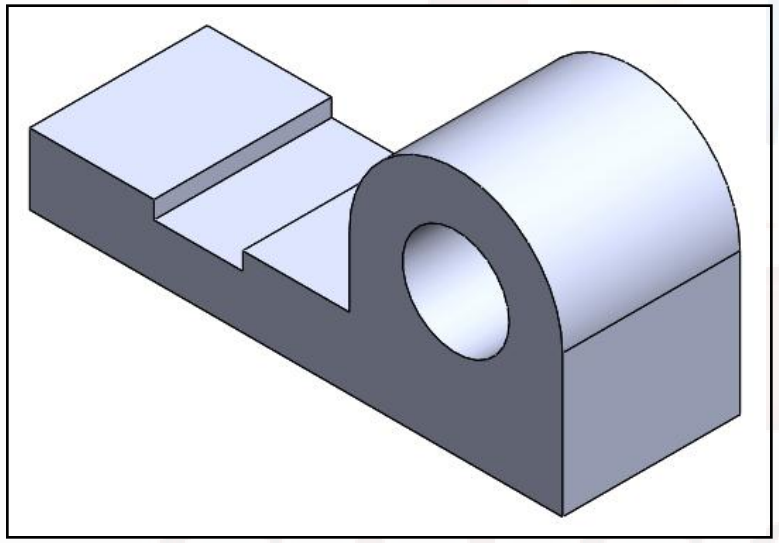

Figure 64 Solid model for Tutorial 3

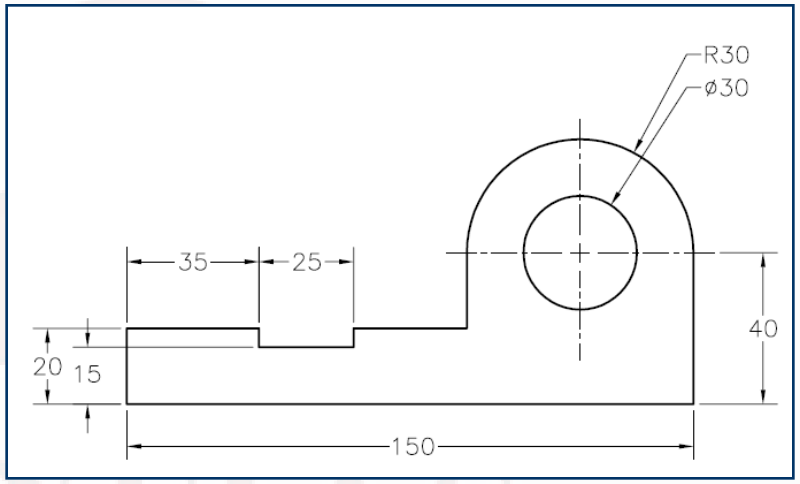

Figure 65 Sketch for Tutorial 3

Tutorial for practice

The following steps are required to complete this tutorial:

1. Start SOLIDWORKS and then start a new part file.
2. Invoke the sketching environment.
3. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5 mm instead of 10 mm.
4. Draw the outer loop of the sketch by using the **Line** tool.

Tutorial for practice

5. Draw the inner circle using the **Circle** tool, refer to **Figure 66**.

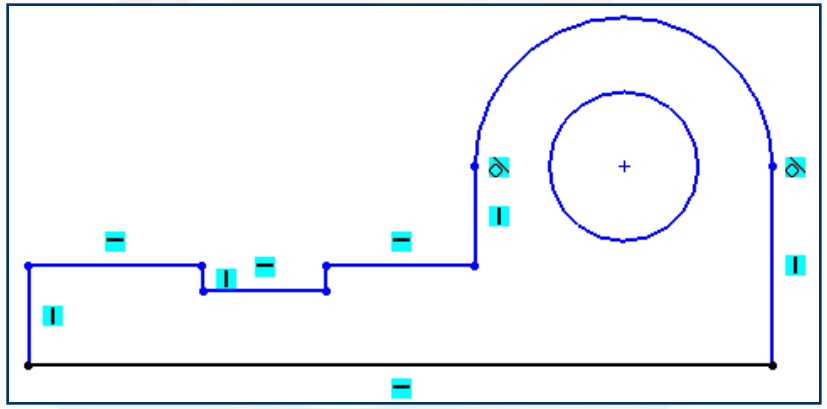

Figure 66 Final sketch for Tutorial 3

6. Save the sketch and then close the file.

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➢ Tutorial 4

In this tutorial, you will draw the sketch of the model shown in **Figure 67**. The sketch of the model is shown in **Figure 68**. Do not dimension the sketch. The solid model and the dimensions are given for your reference only. **(Expected time: 30 min)**

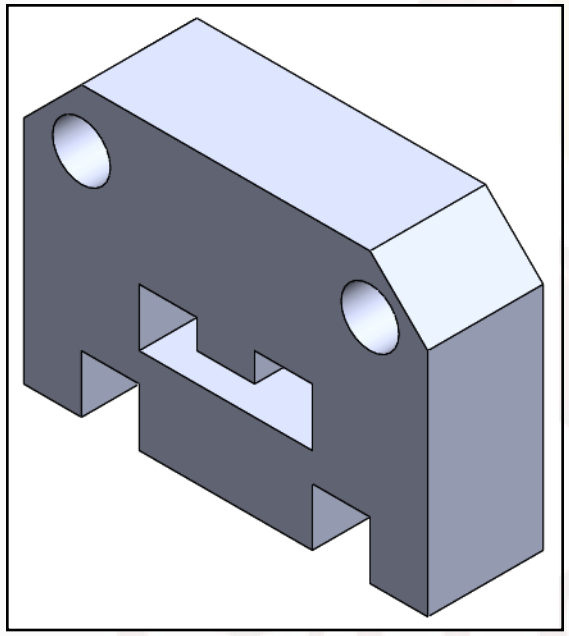

Figure 67 Solid model for Tutorial 4

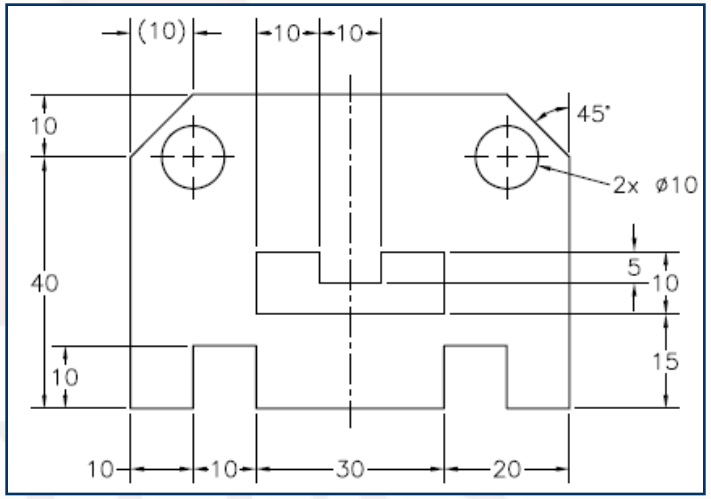

Figure 68 Sketch of the model for tutorial 4

Tutorial for practice

The following steps are required to complete this tutorial:

1. Start SOLIDWORKS and then start a new part document.
2. Switch to the sketching environment.
3. Draw the sketch of the model using the **Line** and **Circle** tools, refer to **Figures 69** through **73**.

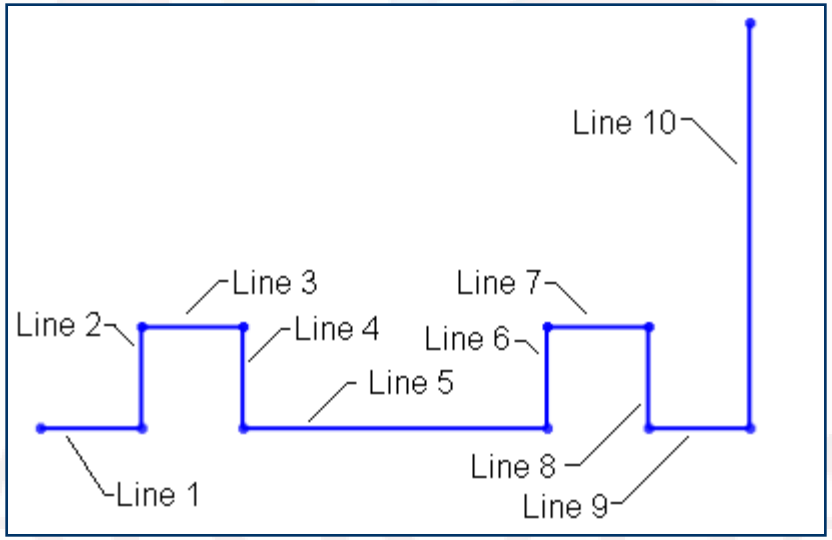

Figure 69 Partial outer loop of the sketch

Tutorial for practice

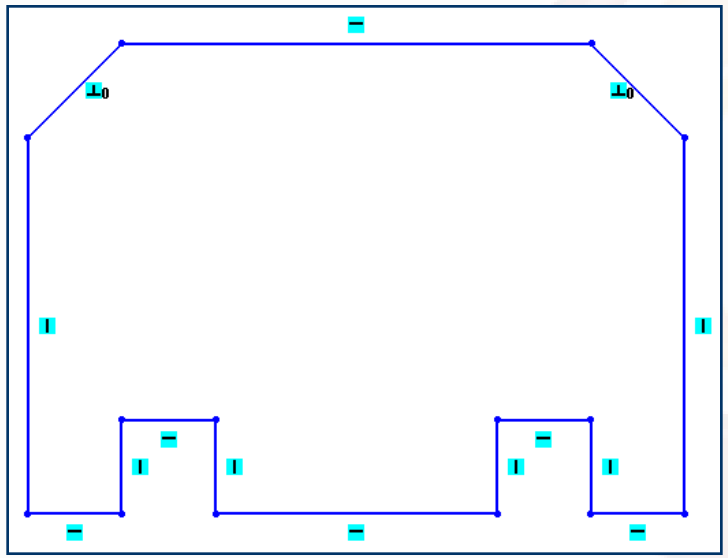

Figure 70 Outer loop of the sketch

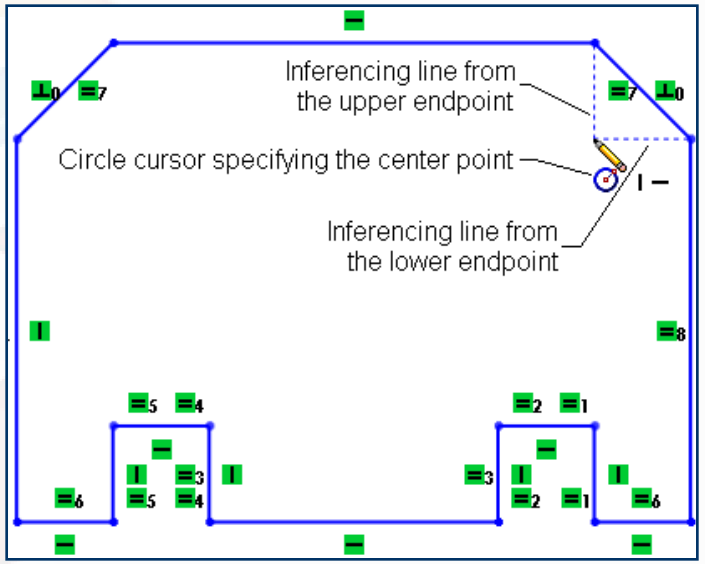

Figure 71 Drawing a circle with the help of inferencing lines

Tutorial for practice

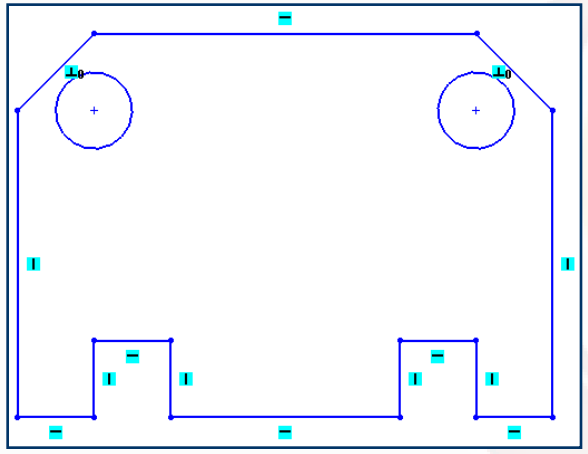

Figure 72 Sketch after drawing the two inner circles

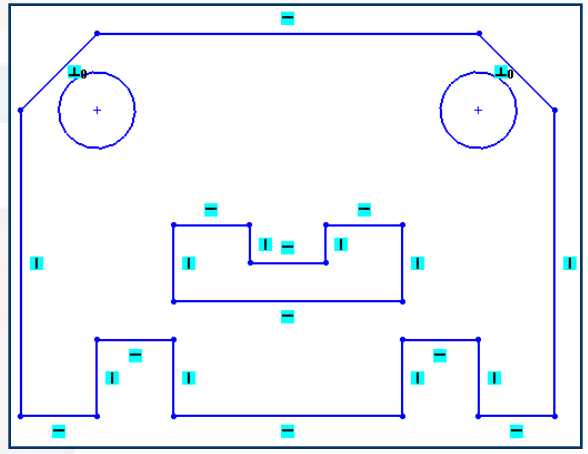

Figure 73 Final sketch for Tutorial 4

4. Save the sketch and then close the document.

Tutorial for practice

Exercise 1

Draw the sketch of the model shown in **Figure 74**. The sketch to be drawn is shown in **Figure 75**. Do not dimension the sketch. The solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

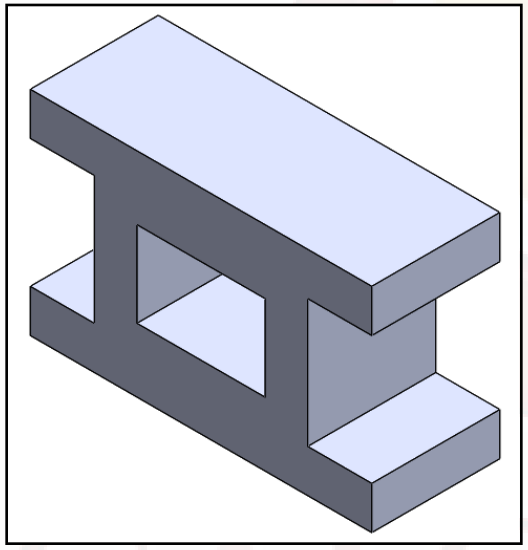

Figure 74 Solid model for Exercise 1

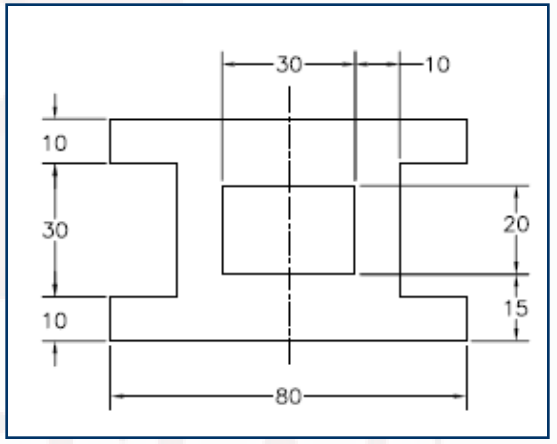

Figure 75 Sketch for Exercise 1

Exercise 2

Draw the sketch of the model shown in **Figure 76**. The sketch to be drawn is shown in **Figure 76**. Do not dimension the sketch. The solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

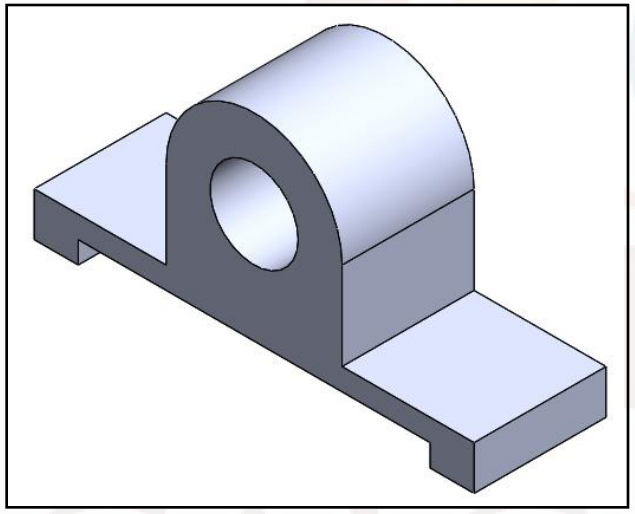

Figure 76 Solid model for Exercise 2

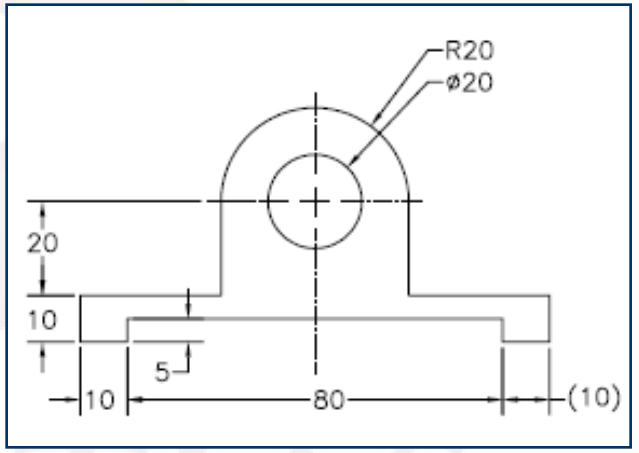

Figure 77 Sketch for Exercise 2

Tutorial for practice

Exercise 3

Draw the sketch of the model shown in **Figure 78**. The sketch to be drawn is shown in **Figure 79**. Do not dimension the sketch. The solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

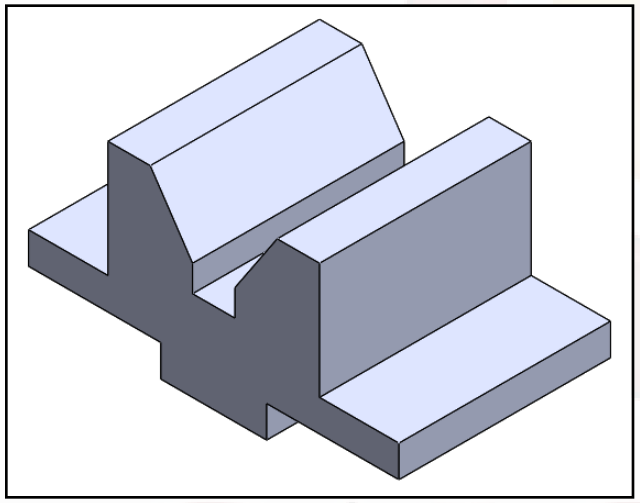

Figure 78 Solid model for Exercise 3

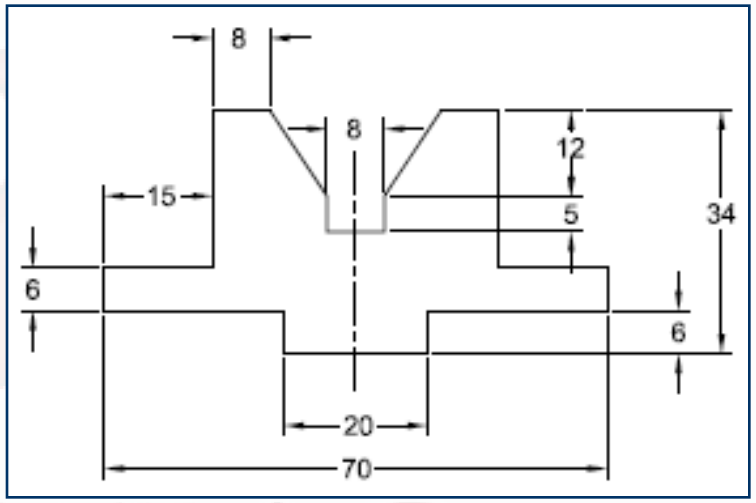

Figure 79 Sketch for Exercise 2

Tutorial for practice

Exercise 4

Draw the sketch of the model shown in **Figure 80**. The sketch to be drawn is shown in **Figure 81**. Do not dimension the sketch. The solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

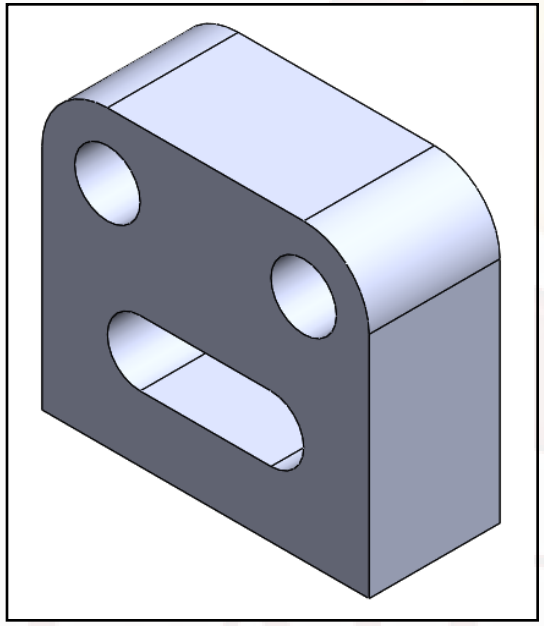

Figure 80 Solid model for Exercise 3

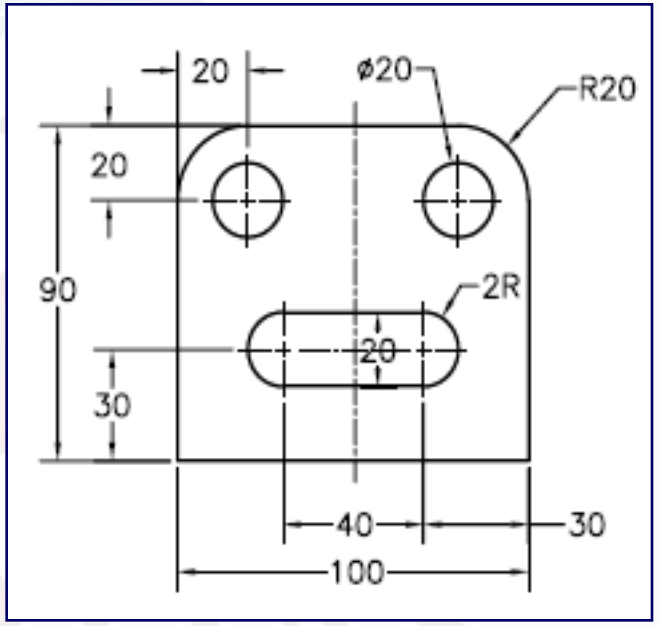

Figure 81 Sketch for Exercise 2

References

- Engineering Drawing by N. D. Bhatt and V. M. Panchal
- Engineering Graphics by K. C. John
- NPTEL

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Thank You

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